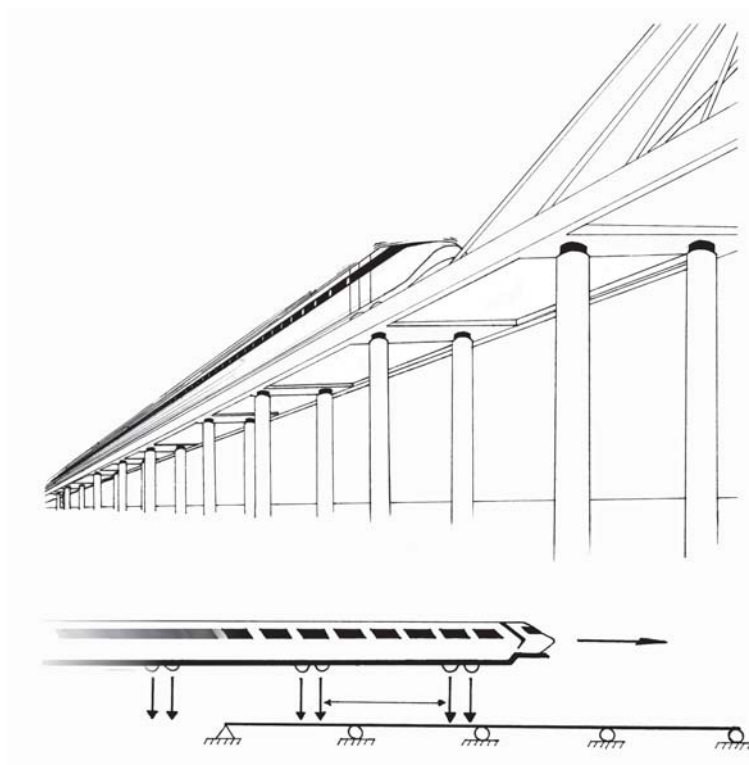LUND
UNIVERSITY

REDUCED MODELS FOR DYNAMIC ANALYSIS OF HIGH-SPEED RAILWAY BRIDGES

ANDREAS GUSTAFSSON

Structural
Mechanics

Master's Dissertation

REDUCED MODELS FOR DYNAMIC ANALYSIS OF HIGH-SPEED RAILWAY BRIDGES

Master's Dissertation by
ANDREAS GUSTAFSSON

Supervisors:

Kent Persson, PhD and Per-Erik Austrell, PhD,
Div. of Structural Mechanics

Morgan Johansson,
Reinertsen AB

Examiner:

Göran Sandberg, Professor,
Div. of Structural Mechanics

ACKNOWLEDGMENTS

ABSTRACT

Today a dynamic analysis is often performed with the use of advanced finite element software. An advanced model normally produces precise results. Analyzing these large models of bridges is however time consuming and a detailed description of the geometry, supports and material properties must be known. In an early stage of a project a reduced model could significantly reduce the time needed to perform the final dynamic analysis. The aim of this thesis was to investigate the dynamic response of railway bridges for high-speed trains and to develop a reduced model that can be used early in a project, to provide a quick dynamic analysis of bridges with several spans.

The reduced model was based on the fact that the fundamental dynamic behavior of certain type of bridges may be described by the dynamic behavior of 2-dimensional Bernoulli beam elements. The bridge model was developed as a MATLAB function and consists of a selectable number of Bernoulli beam elements. Three different approaches of modeling the train load were compared. The first method of modelling the train was made as moving point-loads, the second as a distributed load and the third as a separate mass and spring model.

In order to make the developed MATLAB function easy to use, to perform quick analysis and with good accuracy, the numbers of input parameters were minimized. This was made by setting some of the properties of the reduced model as fixed standard values. The values were based on result of extensive modelling studies.

Finally, two bridges with realistic dimensions and materials were analyzed using both the reduced model and a more advanced model. The reduced model was proved to provide basically the same results as the more advanced model. The time needed to model the bridge and to perform the analysis was approximately 10 minutes when using a standard home pc, which was a significant reduction of the time needed to make the advanced model and to perform the analysis.

CONTENTS

1. Introduction

1.1. Background

Today it is important both from an environmental and financial aspect to reduce our dependence on fossil fuel. An expansion of the railway infrastructure plays an important role in that quest.

In order to be more competitive, compared to other transport modes, many European countries tend to fund the construction of new railway lines for high speed trains. At this moment, Banverket, the authority responsible for rail traffic in Sweden, investigates the efficiency of building high speed railway lines in Sweden and are also planning to increase the velocity on the existing tracks.

The introduction of high-speed trains lead to situations where dynamic loads increases, especially on bridges, and new design methods that take into account high-speed loading. The dynamic behavior of railway bridges is in most design codes traditionally analyzed by following an approach of a so-called single moving load model which is a static load that, multiplied with a dynamic impact factor, represents the increase of a static load to a dynamic situation. This model does not take into account the possibility of resonance effects due to a periodic load.

In 1995, due to several problems with high speed railway lines, in particular excessive high acceleration on bridges that could lead to ballast liquefaction, the European Rail Research Institute (ERRI), decided to establish a committee to study these phenomena. It resulted in new design codes for railways with train-speeds above 200 km/h, which included the necessity to take the effect from periodic load into account.

Performing a dynamic analysis of a bridge, in accordance with the standards, is time consuming and a detailed description of the final bridge parameters must be known. In an early stage of a project a rough estimation of the dynamic behavior of a bridge could significantly reduce the time needed to perform the final dynamic analysis.

1.2. Previous work

The dynamics effects of a single moving load have since the early stage of railways been an important design criteria for the structure. This phenomena have for instance been studied by Timoshenko in [4].

The dynamic effects due to periodic loads have been studied in recent years only. An analytical solution for a simply supported beam subjected to moving point

loads was established by Fryba in [5]. Other work are for instance in [6] and [7]. The Swedish design codes for railway bridges, [1], are based on [2] that is one of several reports published by the European Rail Research Institute (ERRI).

Reduced models have been developed earlier. An example of this is a study carried out by Oscar Jarmillo de Leon and Kasper Lasn in [9]. It resulted in a simplified calculation tool based on a single degree of freedom system. This model does only include the first mode of vibration and is restricted to analyzes for a simple supported beam, with one span.

1.3. Problem statement

Today a dynamic analysis is often performed with the use of advance finite element software. An advance model normally produces precise results. Analyzing these large models of bridges is, however, time consuming and a detailed description of the geometry, supports and material must be known. In an early stage of a project a reduced model could significantly reduce the time needed to perform the final dynamic analysis. To benefit from such a reduced model, it should provide sufficiently accurate results that have to be carefully interpreted.

1.4. Aim

The aim of this thesis was to investigate the dynamic response of railway bridges for high-speed trains and to develop a reduced model that can be used early in a project, to make quick dynamic analyzes of bridges with several spans.

1.5. Scope

The approach of developing the reduced model was defined with the following steps:

• A method that enabled reduced models of railway bridges was developed.
• Based on the reduced model, a MATLAB function aiming at calculating the dynamic response of a bridge, caused by trains passing the bridge was developed.
• The result and the limitation of the program were being thoroughly analyzed and different approaches of modelling the train load were estimated.
• Based on modelling studies the number of input parameters were minimized by setting some of the properties of the program as fixed standard values.
•Finally, the reduced model was verified by comparing the result with a more advance model.

1.6. Limitations

Although the standard specify that several requirement of a dynamic analysis must be examined. In this master thesis the evaluation of the results of the dynamic

simulations were only made by studying deck acceleration and deflection.

1.7. Disposition

In Chapter 2 the theory of structural dynamics is presented.

Chapter 3 describes the design codes that have an influence on the developed design tool.

In chapter 4, first the dynamic phenomena of high-speed railway bridges is described. Thereafter, a reduced model of a railway bridge and a model of the trains are introduced. Finally, dynamic analyzes are performed and the results are analyzed and discussed.

Two other approaches of modelling the train load are described and studied in chapter 5.

In chapter 6, first the final design tool is presented. Thereafter, the tool is verified by comparing the result with a more advance model.

Chapter 7 contains some general concluding remarks and a few proposal for further work.

2. Theory

2.1. General structural dynamic theory

A common situation in structural mechanics is that a structure is only affected by static forces. If the structure is affected by a dynamic force, i.e. a force that is varied in time, it may have a different response compared with the response of a static force.

Properties of a dynamic force, that have essential influence on the structural response, are the amplitude of the force and the relations between the frequency of the load and the natural frequency of the structure, the latter being a structural property that will be described in this chapter. The amplitude of the dynamic force may be so small that it does have a negligible influence on the structural response. In many cases the load frequency is far from the natural frequency of the structure then the force can be expressed as a static force by just applying a dynamic amplification factor to the maximum value of the force. However, in the case where the load frequency is close to the natural frequency such a procedure may not be sufficient and a dynamic analysis may become necessary.

Dynamic forces that are applied on the models may have a large impact on the structure and it will not be sufficient to analyze them as static forces. Structural dynamics describes the behavior of a structure due to dynamics loads and in this chapter some of the basic concept and definitions of structural dynamics are introduced [13].

2.1.1. Equation of motion

The dynamic response of a structure is found by solving the fundamental differential equation of motion. In this section, the equation of motion is derived for a simple structure with a single degree of freedom, so called SDOF.

A structure loaded by a time varying force $f(t)$, causes dynamic responses of the structure, which is described by Newtons second law of motion:

$$\sum F = ma \tag{2.1}$$

F is divided into internal forces that include f_S related to the displacement of the structure and the damping force f_D, related to the velocity of the structure. The forces acting on a structure are illustrated in figure 2.1. Force equilibrium is given below:
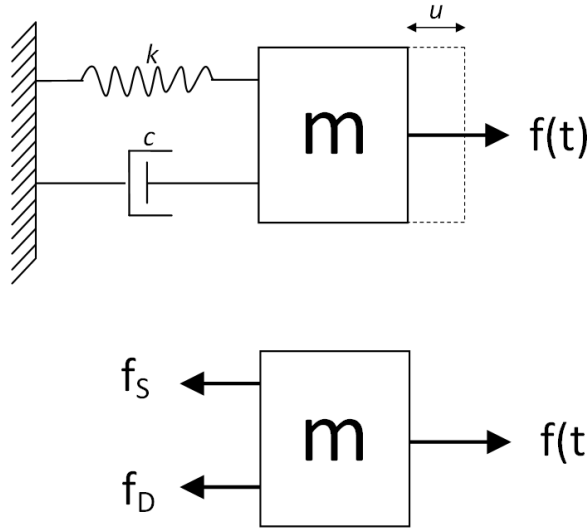
Figure 2.1: *Forces acting on a structure*

$$(\Longrightarrow): f(t) - f_D - f_S = ma \tag{2.2}$$

The displacement, acceleration and velocity can be related to the motion variables as:

$$f_S = ku \tag{2.3}$$

$$f_D = c\dot{u} \tag{2.4}$$

$$ma = m\ddot{u} \tag{2.5}$$

Combining and simplifying equation (2.2) with equations (2.3), (2.4) and (2.5) gives the so called, equation of motion:

$$m\ddot{u} + c\dot{u} + ku = f(t) \tag{2.6}$$

A structure normally has infinite number of degrees of freedom but it is possible to approximate the structure with a system having a finite number of degrees of freedom, a so called multi degree of freedom system, MDOF. Deriving equations of motion for a MDOF-system is similar to the earlier described theory and can be found in, for example in [3]. The equation of motion for a MDOF-system may be written as:

$$\mathbf{M\ddot{u} + C\dot{u} + Ku = F} \tag{2.7}$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{C}$ the damping matrix, $\mathbf{K}$ the stiffness matrix, $\mathbf{F}$ the external load vector and $\mathbf{u}$ the vector of displacements.

2.1.2. Natural frequency and eigenmodes

Normally if a structure is loaded by a dynamic force, the structure starts to vibrate. The vibration signal does normally contain a lot of frequencies. However, a structure deflected in a so called mode shape by a temporary force causes the structure to vibrate with a frequency that corresponds to the specific mode shape. Natural frequencies and natural modes are a vibration property of the structure. A MDOF system has as many natural frequencies, that is to say natural modes, as the number of degrees of freedom of the system.

Resonance appears when the frequency of the loads corresponds to or is a multiple of a natural frequency of the structure. The vibration displacements of the structure is strongly amplified when resonance occurs, but the resonance effect is reduced with the complexity of the mode shape. This means that the lowest natural frequencies, which have a lesser complex shapes, are most important to study if resonance occurs.

The natural vibration properties of the structure are calculated by analyzing the system by free vibration, which means the equation of motion is rewritten without damping, dynamic excitation, external forces and support motion as:

$$\mathbf{m\ddot{u}} + \mathbf{ku} = 0 \tag{2.8}$$

The signal of a natural vibration mode can be described mathematically by:

$$\mathbf{u}_n(t) = \phi_n(A_n cos\omega_n t + B_n sin\omega_n t) \tag{2.9}$$

Where ϕ_n is the deflected shape of the structure and the harmonic function $sin\omega_n t$, describes the time variation of the displacement. A_n and B_n constants determined using the initial conditions of the motion. Combining and simplifying equation (2.8) and (2.9) gives the algebraic function called the matrix eigenvalue problem, which is used to determine the natural mode shapes, ϕ_n, and natural frequencies, ω_n, of the structure [3].

$$[\mathbf{k} - \omega_n^2 \mathbf{m}]\phi_n = 0 \tag{2.10}$$

2.1.3. Numerical solution methods

In order to obtain the response of a structure affected by a time dependent force, the basic equations of motions, described in section 2.1.1, need to be solved. The structure is often too complicated and this fact makes it difficult to establish an analytical solution of the problem. However, approximate solutions can be obtained by solving the equations of motion with the use of numerical methods [6].

Time-stepping methods

There are several methods of solving transient problems using time stepping methods. They are all based on the fact that the applied force may be written as a set

of discrete values, usually at equal time steps. This implies that the equation of motion is to be established and solved at every time-step:

$$m\ddot{u}_i + c\dot{u}_i + (f_s)_i = p_i \tag{2.11}$$

To solve the equations at all time-steps the initial conditions for the displacements, u_0, and velocities, $\dot{u}_0$, have to be known. That information will make it possible, with the use of an appropriate method, to determine the responses u_1 and $\dot{u}_1$ for the first time-step. It is then possible to calculate the acceleration for the next time-step by use of u_1 and $\dot{u}_1$ and carry on iteratively; determining the total response of the structure. To get accurate results it is important that the time-step dt, is short enough. Also, the choice of time-step method is crucial.

A well known time-step method, to be used in this work, is the Newmarks method. It is based on the following equation for displacement and velocity:

$$\dot{u}_{i+1} = \dot{u}_i + [(1-\gamma)\Delta t]\ddot{u}_i + (\gamma\Delta t)\ddot{u}_{i+1} \tag{2.12}$$

$$u_{i+1} = u_i + (\Delta t)\dot{u}_i + [(0.5-\beta)(\Delta t)^2]\ddot{u}_i + [\beta(\Delta t)^2]\ddot{u}_{i+1} \tag{2.13}$$

The parameters γ and β are defining variations of acceleration during a time-step. The choice of parameters is affecting the accuracy and stability of the result [3].

Mode superposition

Mode superposition provides a faster way of solving the differential equation (2.6) with the time-step method. Usually when solving the equations of motion, it also includes solving coupled equations. The mode superposition method implies that the equation of motion can be transformed to a set of uncoupled equations. Moreover, modal decomposition also enables a possibility to reduce the system of equations. The method of mode superposition is based on the fact that the dynamic response can be expressed as a linear combination of the natural eigenmodes of the structure.

$$\mathbf{u}(t) = \sum_{r=1}^{N} \phi_r q_r(t) = \Phi \mathbf{q}(t) \tag{2.14}$$

With the use of modal coordinates, ϕ_n and $q_n(t)$, the transformation to uncoupled equations of motion can be established according to:

$$M_n \ddot{q}_n(t) + K_n q_n(t) = P_n(t) \tag{2.15}$$

Due to the fact that it is often possible to describe an approximate dynamic response with just a few eigenmodes the system equations can often be substantially reduced. In large finite element models a reduction of the system equations saves a lot of computer time. However, excluding modes is a crucial action. It is necessary to select the eigenmodes that have the largest influence on the result.

Therefore, the selected eigenmodes must be chosen carefully and be based on thorough investigations.[3]

2.1.4. Damping of a structure

Damping is the denomination for the reduction of a vibration response. Damping appears, due to energy losses from friction and viscous deformations, in a vibrating structure. It is hard to determine damping properties due to the fact that the damping properties of materials and joining of structure members often are not well established. The damping property has a large influence on the total response of a dynamically loaded structure. It is therefore important that the damping properties of a structure are accurately determined. Often, the best way of determining damping values for a structure is to use data from dynamic tests on similar structures. Otherwise the use of references with typical material values is recommended.

The damping value has proved not to be constant for all modes of vibration and there are different approaches of how to construct the damping matrix $\mathbf{C}$. In this work a so called classically damped system is being used, which is appropriate if the same damping mechanisms are distributed throughout the complete structure. A way to construct the classic damping matrix C, is the use of the Rayleigh damping method. This method is based on the fact that the totally damping of a mode is influenced by both stiffness-proportional damping and mass-proportional damping by:

$$\mathbf{C} = a_0 \mathbf{m} + a_1 \mathbf{k} \qquad (2.16)$$

The values a_0 and a_1 are constants based on the damping of two natural frequencies of the structure [3],[14].

2.2. Numerical calculations

The dynamic response, as described in section 2.1.2, may be written as a combination of the response from all excited modes of vibrations. An exact solution includes an infinite number of eigenomdes. A way of include the impact of all eigenmodes is to establish an analytically expression of the dynamic response.

Once the vibration modes are known, it is necessary to integrate the equation of motion to solve the equation. For this, the response from the dynamic load needs to be established.

Finding an analytical solution of the structural response is often a complicated procedure and not always possible to obtain. Therefore, the use of a numerical method is appropriate. A well-known and established numerically method, that will be utilized in this work, is the finite element method. In this section, parameters in the finite element method that are important to receive accurate results are described [6].

2.2.1. Number of elements and the influence on the result

The accuracy of a finite element (FE) solution is influenced by many factors. An important factor is the choice of element type and size. When modelling a 2-dimensional bridge using beam elements the number of elements in the model are an important parameter. The FE solution get closer to the exact solution when the structure is modelled with more elements. This is due to the fact that fairly simple trial functions are assumed for the beam deflection.[8]

2.2.2. Time step influence on the result

Solving the equation of motion (2.7) by numerical methods, the dynamic response is obtained at constant time-steps. The choice of time-step has an impact on the accuracy of the solution and a very fine time-step is necessary to realistically be able to excite the highest eigenmodes of a structure. According to [3] $\Delta t = 0.1T_n$ would give reasonable accurate results, which implies that ten time increments per period, T_n. T_n is the time-period of the eigenmode with the highest frequency that are considered in the analysis.

3. Design codes

According to the Swedish design codes, instituted by Banverket, a railway bridge that allows speed above 200 km/h has to be calculated concerning dynamic loading. An analysis includes: determination of vertically deformation, rotations close to the supports, torsion and maximum vertically deck acceleration of the bridge. The analysis should be performed using the so called High Speed Load Model (HSLM) in speed intervals from 100 km/h to the maximum speed plus 20 percent. In this chapter the parameters that have an influence on the bridge model used in this work, will be presented.[1]

3.1. Acceleration

In a dynamic analysis of a railway bridge, the deck accelerations has to be calculated and compared with the maximum allowed deck acceleration. An excessive vertical acceleration in a railway bridge associates with loss of ballast stability and other phenomena which have a harmful impact on the structure.

The Swedish code is following the recommendations for maximum allowed deck acceleration issued by ERRI. The present values is set to 3.5 m/s^2 for bridges with a ballast bed and 5.0 m/s^2 for bridges with fasten decks or bridges with slab tracks. These values are based on result from laboratory tests and measurements on actual bridges [1], [2].

3.2. Modes of vibration

A numerical model of a railway bridge has as many modes of vibration as degrees of freedom, but as described in section 2.1.3, by using mode superposition it is possible to reduce the number of modes that are considered to contribute to the dynamic response, which saves a lot of computer time.

The Swedish codes prescribes that only the modes up to 30 Hz should be considered when calculating the maximum vertical acceleration. On the other hand, ERRI specifies that all modes of vibration up to 1.5-2.0 times the eigenfrequency of the first mode must be considered. That is to say that if for instance the first mode of vibration has an eigenfrequency of 35 Hz, all modes of vibration up to 45-60 Hz should be included in the calculations according to ERRI recommendations, compared with the Swedish code which indicates that no mode of vibration have to be considered.

Due to the fact that the Swedish code differ from ERRI:s recommendations the effect of excluding eigenmodes will be carefully analyzed in this work [1], [2].

3.3. Damping

The values for damping specified by the Swedish code are the same as recommended by ERRI. The values differ for different material and are based on research performed by ERRI.The recommended damping values can be found in table 3.1 [1], [2].

Table 3.1: *Values of damping for various spans and type of bridges.*

Bridge type	ξ Lower limit of damping (percent)	
	Span < 20m	Span > 20m
Steel and composite	$\xi = 0.5+0.125(20\text{-}L)$	$\xi = 0.5$
Prestressed concrete	$\xi = 1.0+0.07(20\text{-}L)$	$\xi = 1.0$
Reinforced concrete	$\xi = 1.5+0.07(20\text{-}L)$	$\xi = 1.5$

3.4. High Speed Load Model

The design codes instituted by Banverket, demands that a dynamic analysis should be performed by using a so called high speed load model (HSLM), which are given from idealizations of actual high-speed trains. The weight of a moving train is represented by point loads moving over a bridge model with a predefined distance that are based on the distance between the train axles on standard high-speed trains.

There are two different sorts of high speed load models, HSLM-A and HSLM-B. HSLM-B is a load model representing a single train, which is to be used for bridges with a span shorter than 7 meters. Bridges having a span that exceeds 7 meters should be analyzed using HSLM-A, which consist of 10 different load models, A1-A10. Data for each respectively load model is shown in table 4.1 and an illustration of the trains can be found in figure 3.2.

A complete dynamic analysis of a high-speed train bridge, using HSLM-A, includes the calculation of dynamic response for all ten load models individually, at velocities from 100 km/h to the maximum speed of the train plus 20 percent, in steps of 2.5 km/h [1].
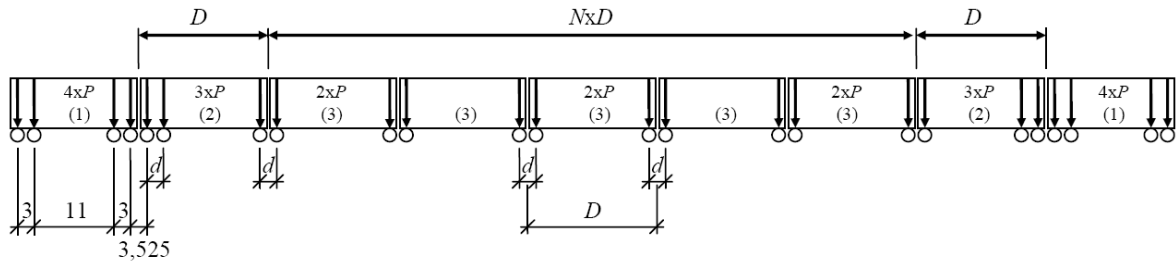
(1) Power car (leading and trailing cars identical)
(2) End coach (leading and trailing and coaches identical)
(3) Intermediate coach

Figure 3.1: *The load distribution for train model HSLM-A.*

Table 3.2: *Train configurations for train HSLM-A.*

Train model	Number of intermediate coaches	Coach length	Bogie axle spacing	Point force
	N	D [m]	d [m]	P[kN]
A1	18	18	2.0	170
A2	17	19	3.5	200
A3	16	20	2.0	180
A4	15	21	3.0	190
A5	14	22	2.0	170
A6	13	23	2.0	180
A7	13	24	2.0	190
A8	12	25	2.5	190
A9	11	26	2.0	210
A10	11	27	2.0	210

4. Modelling railway bridges using 2-dimensional beams

In this chapter, first the dynamic phenomena of high-speed railway bridges is described. Thereafter, a reduced model of a railway bridge and a model of the trains are introduced. Finally, dynamic analyzes are performed and the results are analyzed and discussed.

4.1. Dynamic behavior of high-speed railway bridges

The risk that resonance frequencies are excited in railway bridges are increased when the velocity of the trains that passes the bridge exceeds 200 km/h. This phenomena appears when the frequency of the train loads or a multiple of the frequency matches a natural frequency of the bridge. The vibrations amplitudes in the bridge caused by the load from train axles are strongly built up when resonance effect appears and it is important that the bridge is constructed to withstand that phenomena [2].

Figure 4.1 shows a relation between the first natural frequency and the span length of railway bridges. The results are based on a dynamic research on 113 railway bridges presented in [5].

The load frequency of a train is dependent on the distance between equally spaced train axles and the velocity of the train, according to:

$$f = \frac{v}{D} \tag{4.1}$$

A schematic illustration of a train and the characteristic length D between axles is shown in figure 4.2.

To illustrate what happens when resonance occurs, consider a 10 meter simple supported beam with the first natural frequency equal to 7 Hz. Several point loads with a characteristic distance D = 17 meter, are passing the beam with a constant velocity v. When v = 450 km/h the load frequency, according to equation 4.1, is equal to 7 Hz and resonance occurs. When the velocity is equal to a multiple of the bridge frequency, in this case when v = 225 km/h, which gives a load frequency of 3.5 Hz, resonance was also appearing. The dynamic response obtained at the center of the beam for three velocities (v = 500 km/h, v = 450 km/h and v = 225 km/h) are shown in figure 4.3.

1 – Steel truss bridges
2 – Steel plate girder bridges with ballast
3 – Steel plate girder bridges without ballast
4 – Concrete bridges with ballast
5 – Concrete bridges without ballast
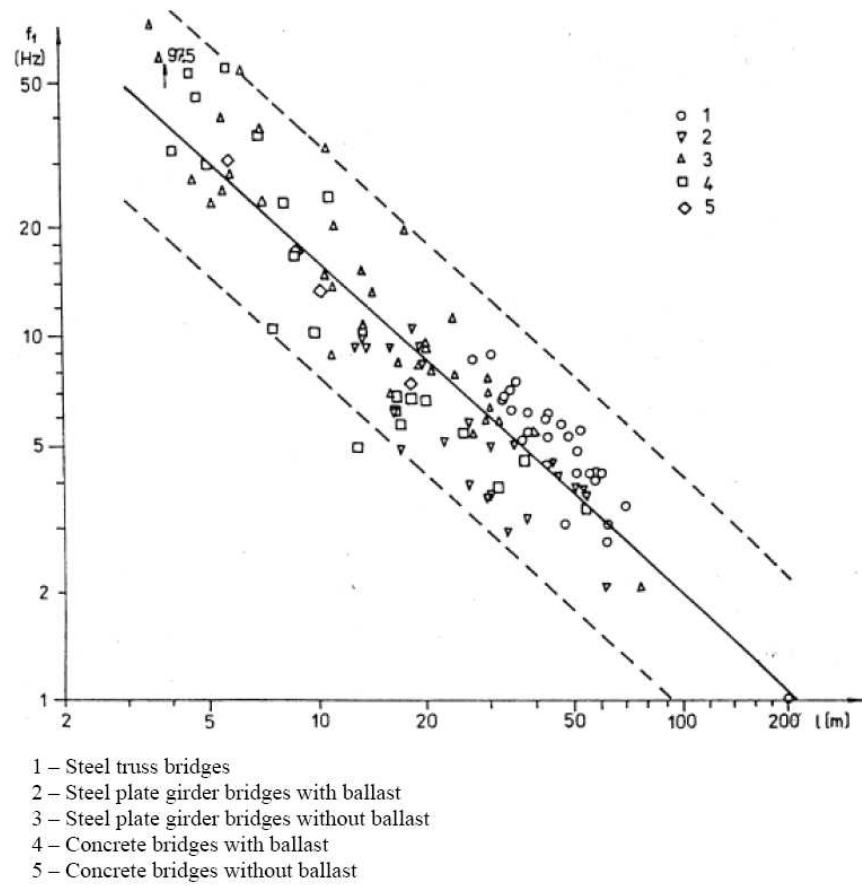
Figure 4.1: *First natural frequency from 113 measured railway bridges.*
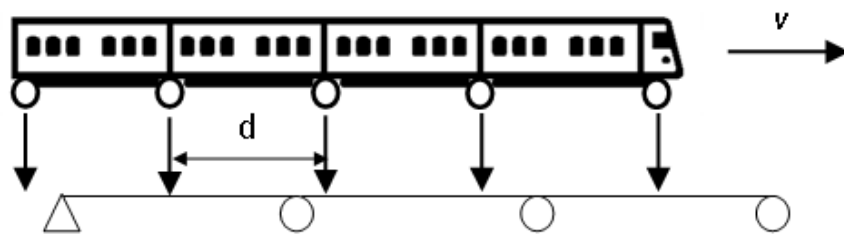
Figure 4.2: *A schematic illustration of a train*
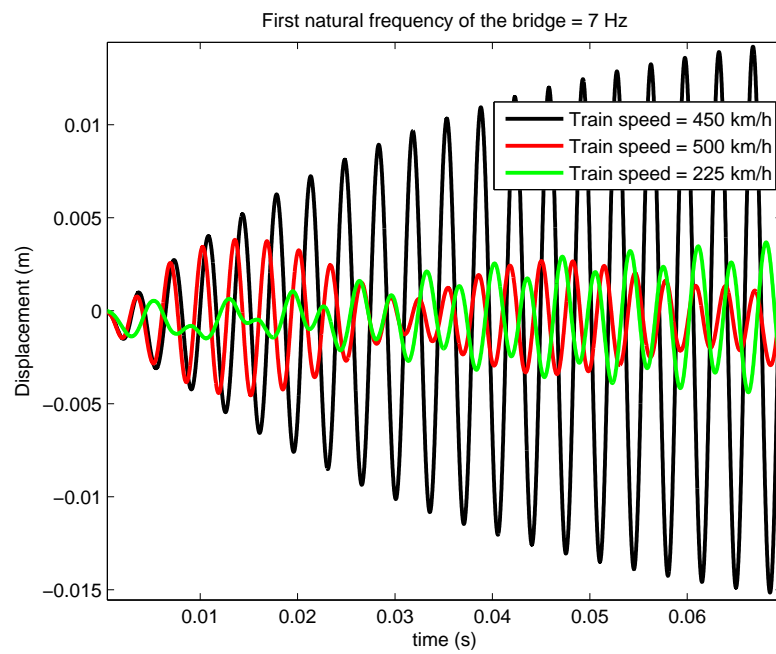
Figure 4.3: *Displacement in a point in the center of the bridge.*

4.2. Bridges modelled by beams

A dynamic analyzes of railway bridges are often performed using large 3-dimensional finite element models. An advance model normally produce precise results. Analyzing these large models of bridges are, however, time consuming and a detailed description of the geometry, supports and material must be known.
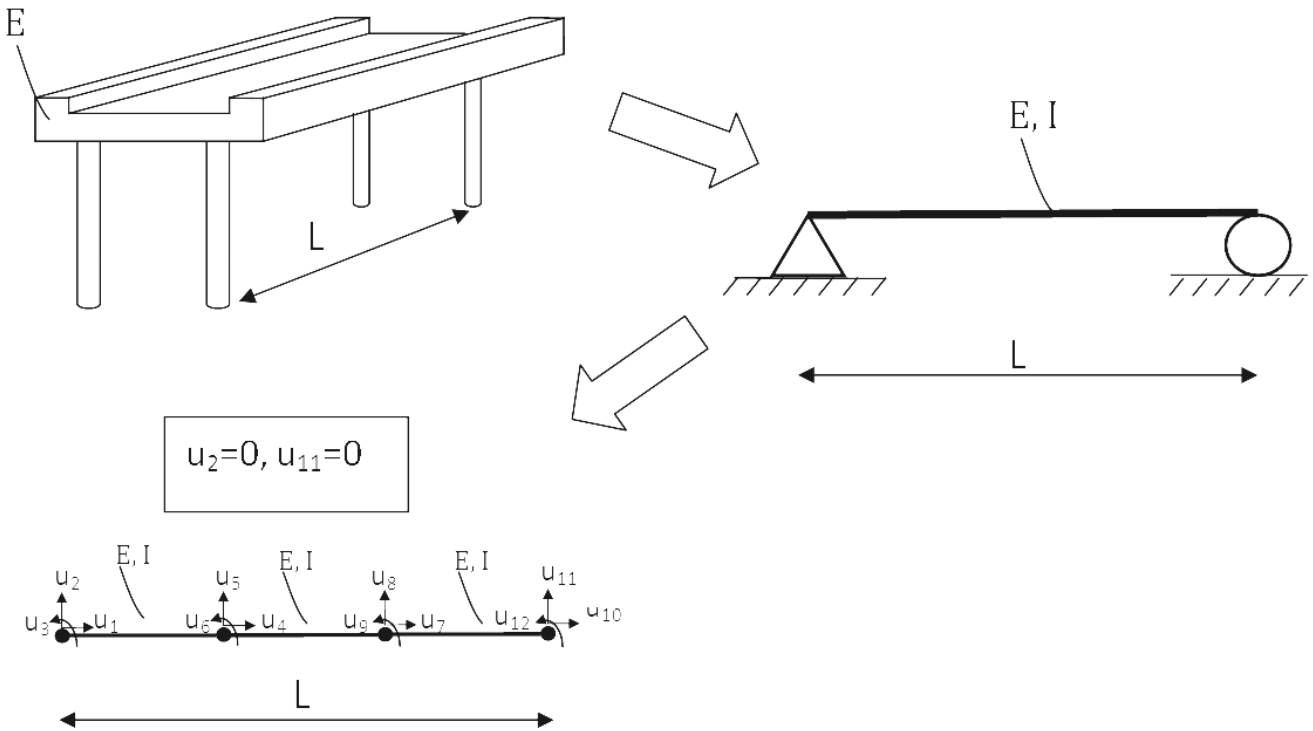
Figure 4.4: *Transformation of a bridge to a reduced beam element model.*

In this work, the dynamic response of railway bridges were analyzed using reduced models of train bridges. The models were based on the fact that the fundamental dynamic behavior of certain type of bridges may be described by the dynamic behavior of 2-dimensional Bernoulli beams. The Bernoulli beam was modelled using varied number of beam elements. A schematic illustration of the transformation of a railway bridge to a beam element model is shown in figure 4.4.

In this section the assumptions, construction and limitations of the model is described.

4.2.1. Assumptions and limitations of the model

• 2-dimensional beam models can only calculate vertically and horizontally bending modes. A key assumption in this work was therefore that vertically modes of vibration contributes to the vertically accelerations of the bridge, which implies that it was assumed that accurate results may be achieved even though torsional and

horizontal bending modes are neglected. This was a limitation of the model since all bridges have torsional bending modes and, if they are excited, they often increase the vertically acceleration of the bridge. Torsional bending modes are mainly excited when a bridge is subjected to eccentric dynamic loads, as on 2-rail bridges, and the reduced model was therefore more accurate for bridges with a centric track.

• It was also assumed that the frequencies and shapes of the modes that were calculated using the reduced model were nearly the same as the frequency and eigenmodes of the bridge.

• The bridge was modelled using Bernoulli beams, which implies that shear deformation was neglected. When modelling truss bridges shear deformations can not be neglected. Therefore, the reduced model was limited to non truss bridges.

• The value of the moment of inertia of beams was assumed to correspond to the cross-section of the bridge but only the supporting structure it-self, not rail, ballast and sleepers.

• The Young's modulus, E, is a material parameter and its value is found in the design codes. Therefore, it was assumed that those values also could be used for the Bernoulli beams.

• The mass used for the bridge model was assumed as the total weight of the complete structure. That was to say; the weight from the rail, ballast, sleepers, handrail and the supporting structure it-self.

• Another limitation of the model was that the beams all have the same properties. To make benefit of the result using the reduced model, it may be limited to railway bridges that have constant stiffness and weight along the length.

• A last assumption was that columns and foundations were assumed to have a neglected vertical deflection and were therefore modelled as simple supports.

4.2.2. The bridge model

The reduced bridge model consists of a selectable number of beam elements and the loads of the train are modelled as moving point loads.

The model was developed as a MATLAB function called *mainprogram*. The properties of the beam elements such as the damping value, Young's modulus, moment of inertia and mass per unit length were input parameters in *mainprogram*. The supports of the bridge was modelled by assigning the displacement in the element nodes to zero and the position of the supports position as input data to *mainprogram*. The number of included eigenmodes in an analysis and the number of time steps per element was also inparameters to *mainprogram*.

CALFEM is a Matlab toolbox for finite element applications and is described in [11]. The functions in CALFEM are executed with specific command lines and several function have been used in this work.

In this section the development of the program is presented.

Setting up the stiffness, mass and damping matrices

The following steps were preformed to set-up the stiffness, mass and damping matrices:

- Creating the matrices Ke, Me and Ce for a 2D elastic Bernoulli beam element.
- Developing a topology matrix based on the bridge dimension and the number of elements.
- Assemble the element matrices Ke, Me and Ce in the global matrices **K**, **M** and **C**.

For a more detailed description see the program code in appendix A.

Constructing the load vector

Figure 4.5 illustrates a point load P that moves over a bridge consisting of three beam elements. The force moves with the velocity v between and over the nodes of the model. In a finite element model all loads are applied at the element nodes. The load P is therefore assigned as equivalent nodal forces, when is situated between the nodes of a beam element. The equivalent shear forces V and moments M are calculated as:
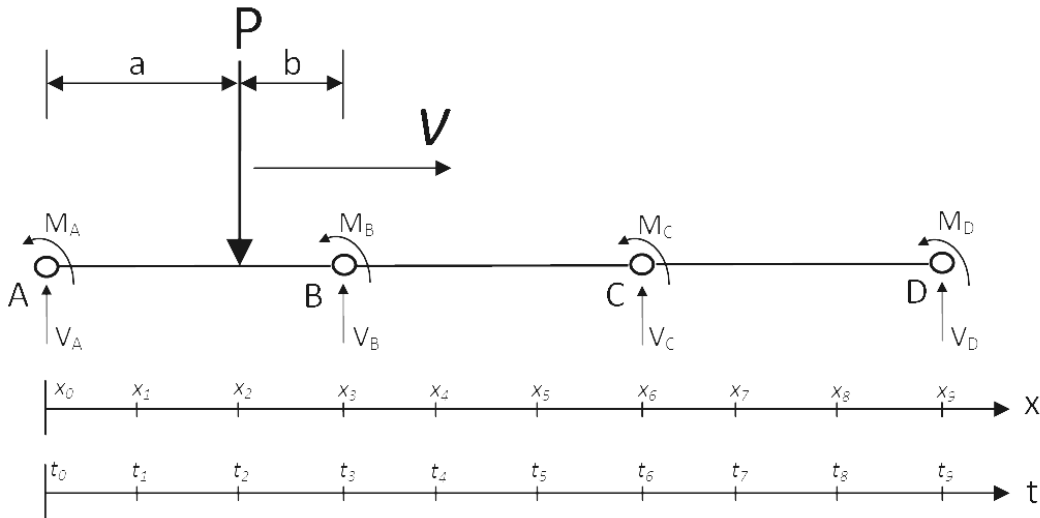
Figure 4.5: *The equivalent nodal forces of a point load on a beam element.*

$$V_A = \frac{Pb^2}{L^2}(1 + \frac{2a}{L}) \tag{4.2}$$

$$M_A = \frac{Pab^2}{L^2} \tag{4.3}$$

$$V_B = \frac{Pa^2}{L^2}(1 + \frac{2b}{L}) \tag{4.4}$$

$$M_B = -\frac{Pa^2b}{L^2} \tag{4.5}$$

The **F** matrix describes the load history at each node of the model for the whole time period of the analysis. The matrix was constructed by calculating the equivalent node loads using equations (4.2)-(4.5) every time the load P moves a distance $\triangle$x. The size of the distance depends on the velocity v and the size of a time step $\triangle$t according to:

$$\triangle x = v \cdot \triangle t \tag{4.6}$$

According to the design codes, a train must be modelled using point loads, where the number and the size of the point loads are varied. The distance between the point loads were also varied. The values that were used when doing a dynamic analysis can be found in the high speed load model, described earlier in chapter 3.

The F matrix in the model was developed as a MATLAB function that needed the following input data:

- Length of the bridge
- Number of element
- Number of time-steps per element
- Velocity of the point loads

For a more detailed description see the program code in appendix A.

Solving the equations of motion

After constructing the **K**, **C**, **M** and **F** matrices the dynamic responses in the bridge could be calculated by solving the equations of motion (2.7). This was performed by using Newmarks time-step method, described in chapter 2.

4.3. Results and analysis
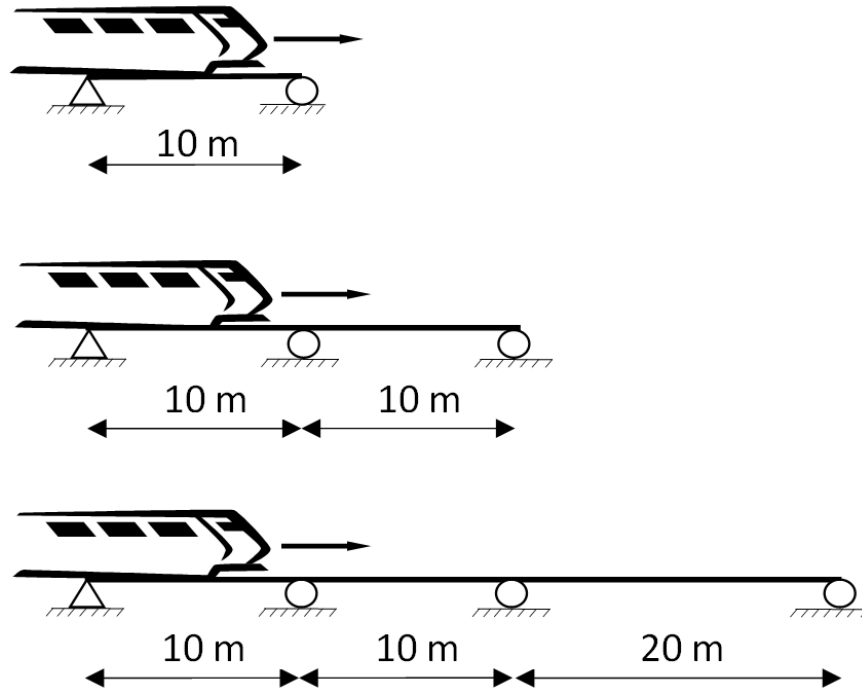
Figure 4.6: *Bridge 1, bridge 2 and bridge 3.*

In order to estimate the influence that some of the model parameters had on the result of a dynamic analysis, three bridges were studied. The three bridges that were studied differed in length and number of supports. The first bridge, denoted Bridge 1, had a length of 10 meters and a simple supported at its end. The second bridge, denoted Bridge 2, had a length of 20 meters and had two spans, each of 10 meters. The third bridge, denoted Bridge 3, had a length of 40 meter and had three spans. The two first were 10 meters and the last was 20 meter. All the bridges had the same E-module, stiffness and mass per meter.
In this section the results of the analyzes of the bridges are presented.

4.3.1. Modelling the initial load

In figure 4.7 it is shown that a point load that was modelled with two different methods, instant loading and preloading gives very different results. Instant loading means that the force affecting the bridge varies from zero to the value of the point load in one time-step. Using the preloaded methods the value varies from zero to the value of the point load in several time-steps before the point load starts to move over the bridge. Instant loading proved to cause very high initial acceleration. The acceleration value was in some cases even proved to exceed the maximum acceleration achieved when several point loads at equal distance passed the bridge at resonance speed. This is shown in figure 4.8. A point load that was modelled as

preloaded does not cause high initial accelerations. The high initially accelerations that appear when the point load was modelled with an instant loading appears since the frequency content is high. Since the method of preloading was shown to provide better results it was used throughout this work.
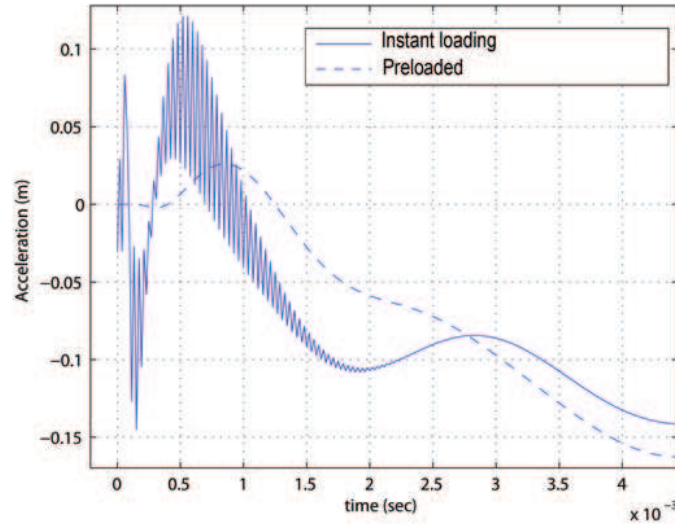
Figure 4.7: *The acceleration as a function of the time.*
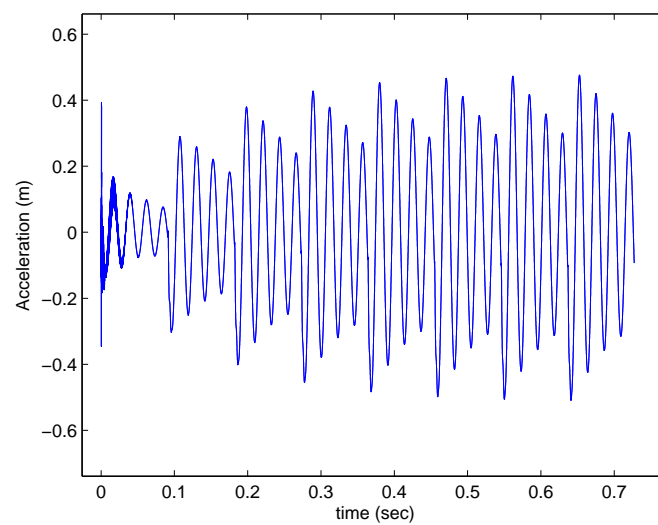
Figure 4.8: *Modelling initial load modelled with the instant loading method.*

4.3.2. Position for maximum accelerations and maximum displacements

In order to determine how the maximum acceleration and maximum displacement may vary along the length of the bridge, the three bridges in figure 4.6 was studied. The calculations are performed for HSLM A1,A2 and A3, described in chapter 3, for a range of velocities from 100-300 km/h in steps of 3.6 km/h. The results are shown in figure 4.9 and presented as the maximum acceleration and displacement at each element node.

The results show that the position of where the maximum acceleration appear in the bridge, is not at the same position as the maximum displacement. The acceleration of a simple supported beam is, for example, not maximum at the center of the bridge. The results also indicate that the actual span of where the maximum acceleration appeared may vary for different load models.

Thus, it is not obvious where in the bridge the maximum values was expected to be found. It is important when analyzing a bridge, that the accelerations points over the whole bridge was to be analyzed. Due to the fact that values for accelerations only can be calculated in element nodes, it was important to model the bridge using an adequate number of beam elements.
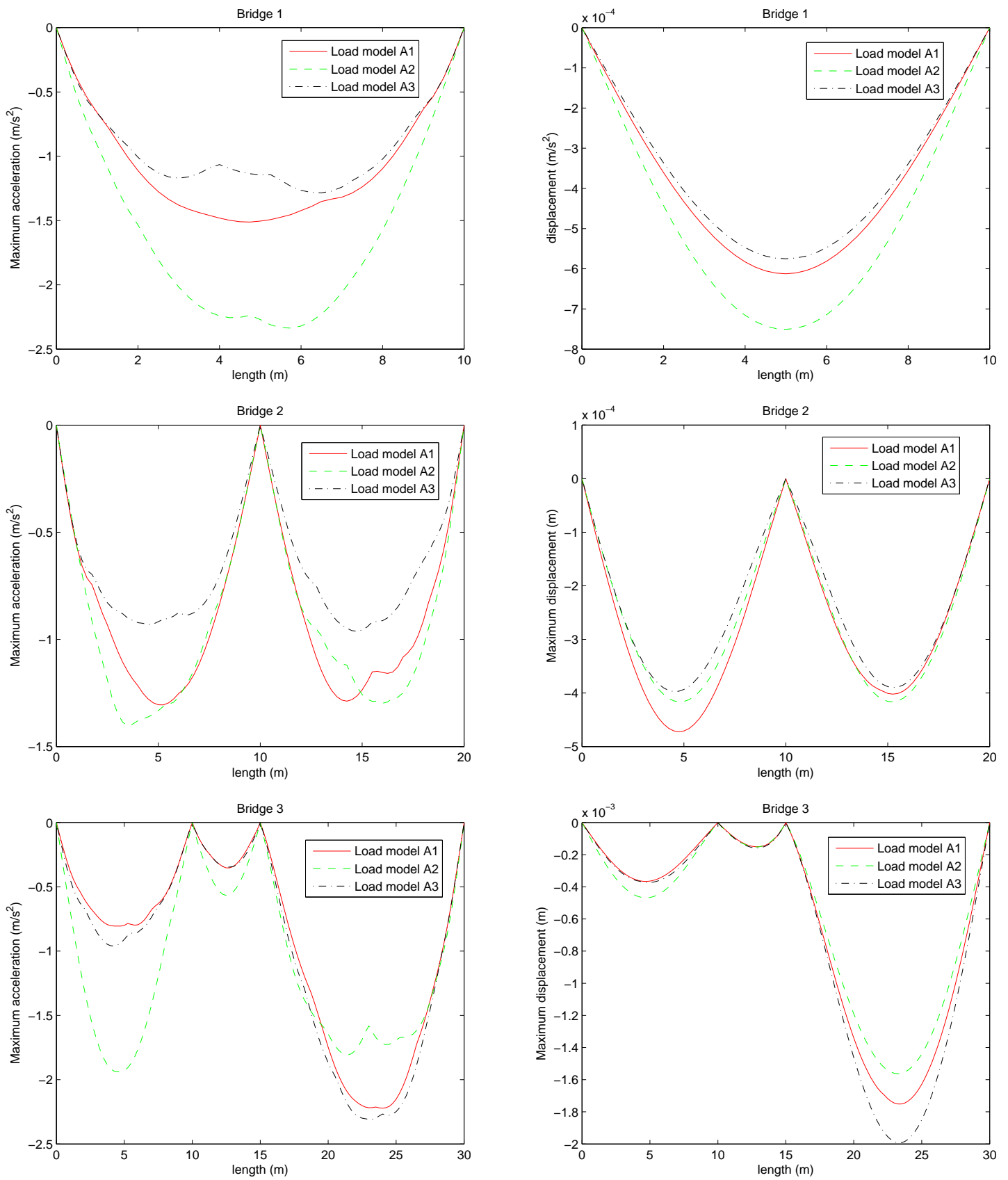
Figure 4.9: *The maximal acceleration and displacement in bridge 1,2 and 3.*

4.3.3. Number of elements and the influence of time-step on convergency

In order to estimate the influence of the number of beam elements and the number of time-steps per element on the results, the bridges presented in figure 4.6 was studied.

In figure 4.10 the maximum acceleration as a function of the number of elements used in the bridge model is shown. The maximum acceleration produced by HSLM-A1, for a range of velocities from 100-300 km/h in steps of 3.6 km/h for all three bridges was plotted in figure 4.10.

In order to investigate the effect of the time-step, the bridges was modelled using 10, 30 and 50 number of time-steps per element. The results for the three choices are presented in each figure. A plot of the value obtained when using 200 elements and 1000 time-steps per element is presented as the exact value and is also shown in the figure.

The results shown in figure 4.10 reveals that the maximum acceleration in a simple supported bridge with one span are getting close to the exact value when the bridge was modelled using 10 elements. It is obvious that the calculated value is getting closer to the exact solution when more time-steps per element were used, but the computer time needed to perform the calculation also increased. The results does also show that it is a clear difference in the result when the bridge was modelled with an element node placed exactly in the center of the span. That is to say when the bridge was modelled using equal number of elements.

The results in bridge 3 and 2 in figure 4.10 show no obvious difference when the number of time-steps per element were varied. It may be observed that, the number of elements needed to obtain a solution close to the exact solution was not the same for both bridges. Bridge 2 gets close to the exact solution when more than 10 elements were being used. Bridge 3, that was longer and had more supports, needed to be modelled using approximately 60 elements to obtain a good solution.

The study of the results implies that the number of elements needed to model a bridge depends on the length and the complexity of the bridge. Varying the number of time-steps per element only had an obvious impact on bridge 1. On bridge 1 approximately the same results were obtained by using 50 time-steps per element, as by using more elements and less time-steps per elements. When reducing the number of time-steps and increasing the number of elements the model was more efficient, considering the computer-time needed to perform the analysis.

The MATLAB program was developed in such a sequence that the element mesh was created without considering the positions of the supports. Therefore, the supports was placed in the element nodes as close as possible to the desired position. This fact had an influence on the number of elements needed to model a bridge precisely. If for instance a bridge was modelled using 4 elements per meter, it implied that supports could only be placed exactly at 4 positions per meter. If a support is to be placed in for instance position 5.60 meter the bridge must be modelled using more than 4 elements per meter, otherwise the support is placed in position 5.50.
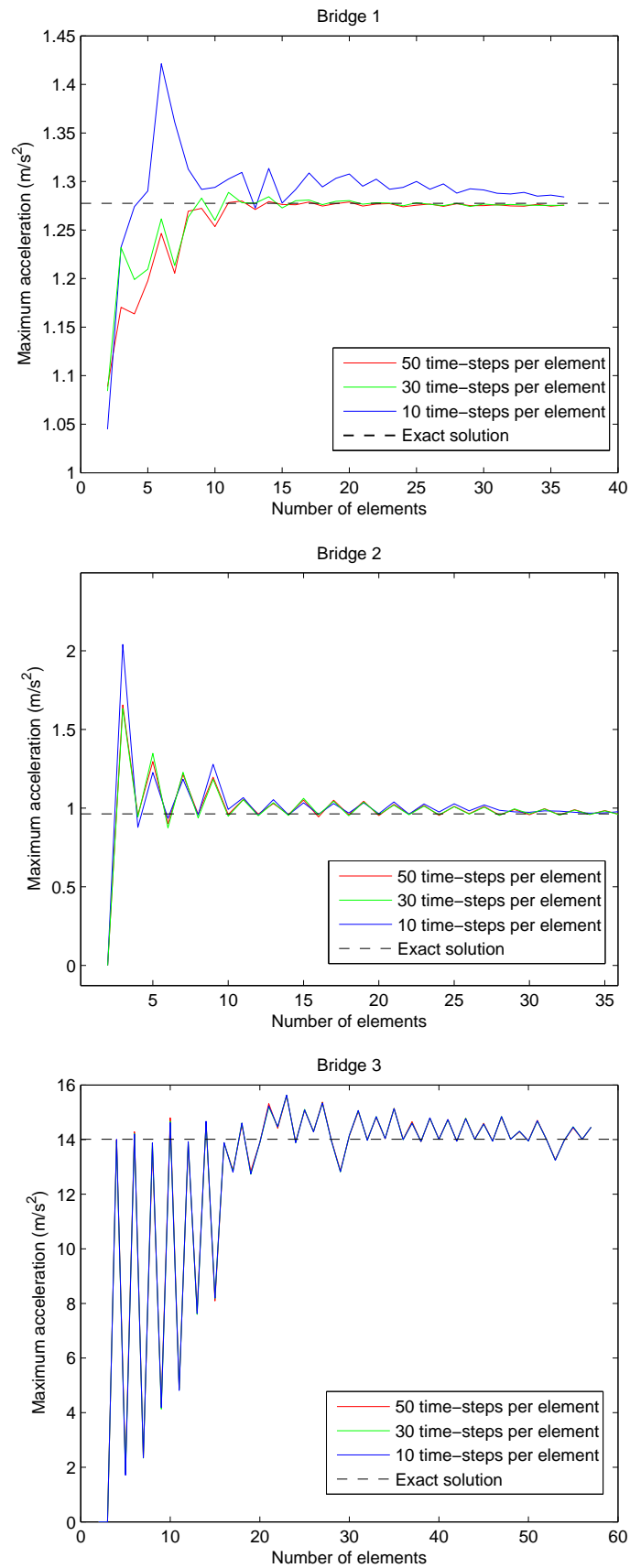
Figure 4.10: *The maximal acceleration as a function of the number of elements in bridge 1,2 and 3.*

4.3.4. Number of included eigenmodes and its influence on convergency

As described in section 2.1.2, the number of included eigenmodes have an influence on the result of a dynamic analysis. In order to estimate the influence the shape of a bridge had on the number of eigenmodes needed to obtain an accurate result, the three bridges described earlier were again studied for various number of eigenmodes.

In figure 4.11, the first four eigenmodes of the three bridges are shown. Figure 4.12 show graphs that describe the maximum acceleration as a function of the length of the bridge. Each plotted value for maximum acceleration was produced by train set A1, for a range of velocities from 100-300 km/h in steps of 3.6 km/h. The dashed graph shows the result obtained from a model where all eigenmodes contributing to the dynamic response. The other graphs shows the response when one, two,three or four number eigenmodes were considered. The results obtained when only one mode was considered is found at the right side in figure 4.12.

The results showed, as expected, that the vertical mode had no influence on the maximum vertical acceleration. Except for that, the calculated maximum accelerations came closer to the exact solution when more modes were included in the analysis. The first mode of vibration had the greatest contribution to the dynamic response for all three bridges as shown in figure 4.12. However, the results prove that the maximum value in all cases were underrated when only the first mode was considered. When the first four modes were included, the maximum peak was close to the maximum peak of the exact solution. It could also be observed that the dynamic response of the first span of bridge 3, was not accurate when only the first mode was considered due to the fact that the third and second mode contributed to the most of the dynamic response.

The computer-time needed to perform an analysis when one mode was included compared with the time when four modes were included was negligible. Based on these results at least four modes should be included when performing an analysis. However, in the design codes, described in section 3.2, the criteria for performing a dynamic analysis was prescribed to only consider the modes with a frequency below 30 Hz. That fact implies that only the first mode should be included in the analysis for bridge 1 and 2 while bridge 3 should include the three first modes.
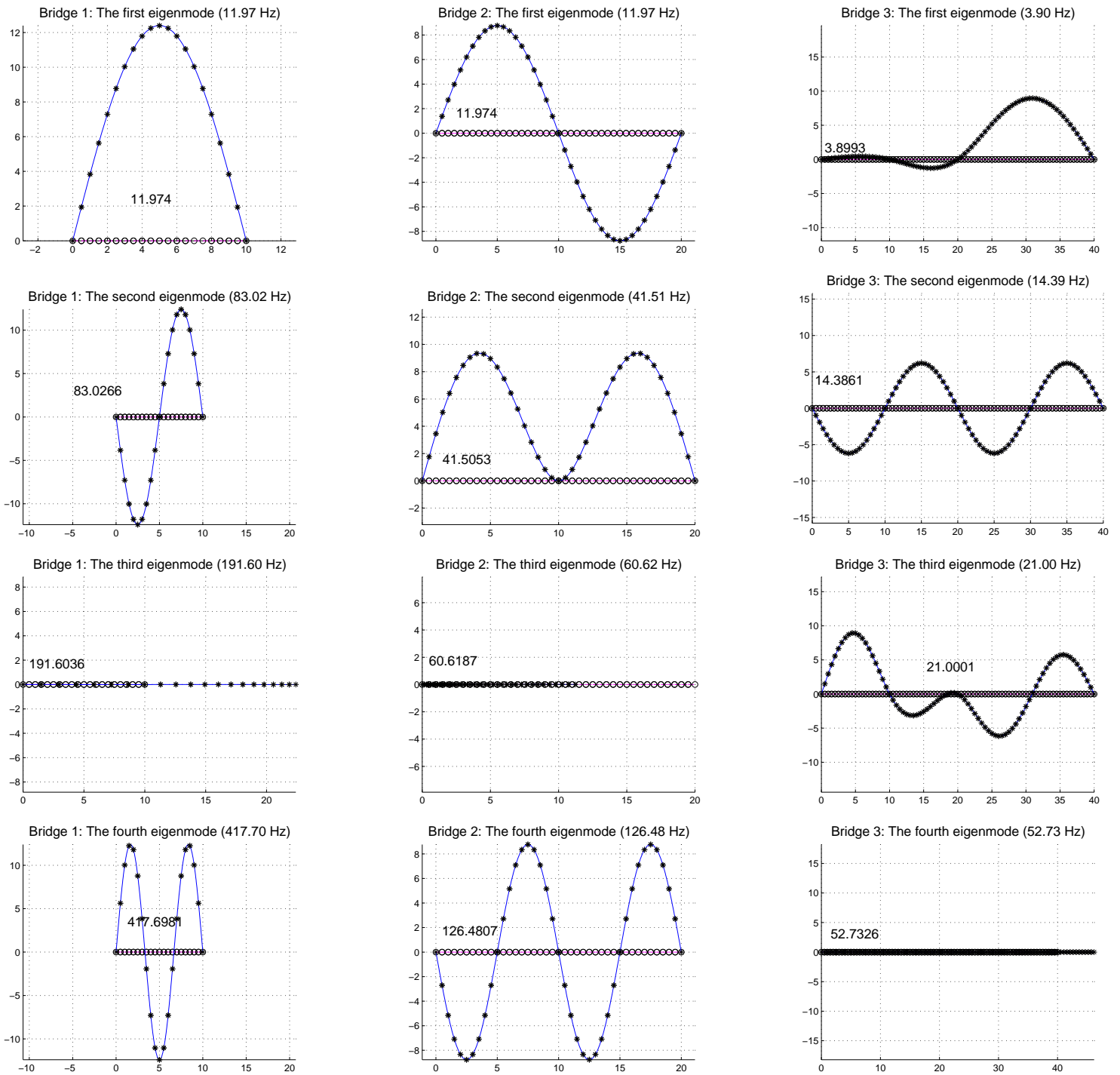
Figure 4.11: *The four first modes of vibration for bridge 1,2 and 3.*
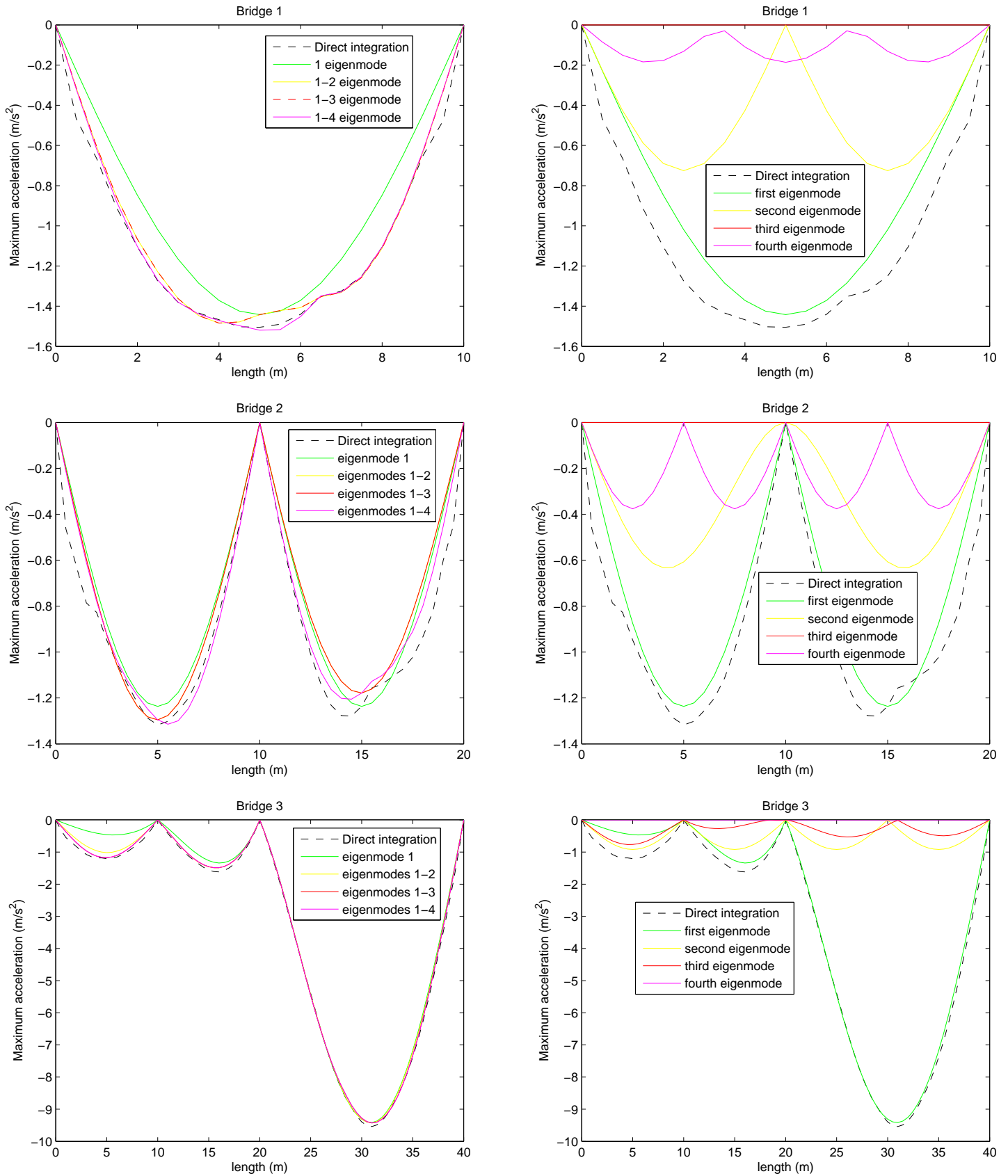
Figure 4.12: *The maximum acceleration obtained including a varied number of eigen-modes for bridge 1,2 and 3.*

4.3.5. Acceleration velocity plot

A way to visualize the result of a dynamic analysis is to plot the maximum acceleration in the bridge deck as a function of the speed of the train. By studying the results it is easy to evaluate the dynamic impact on the bridge and the velocity range that contributes to the most dynamic response. Figure 4.13 shows a dynamic/velocity plot for bridge 1, 2 and 3 produced by train set A1-A10. The result shows at what velocity Bridge 1, Bridge 2 and Bridge 3 have resonance peaks.
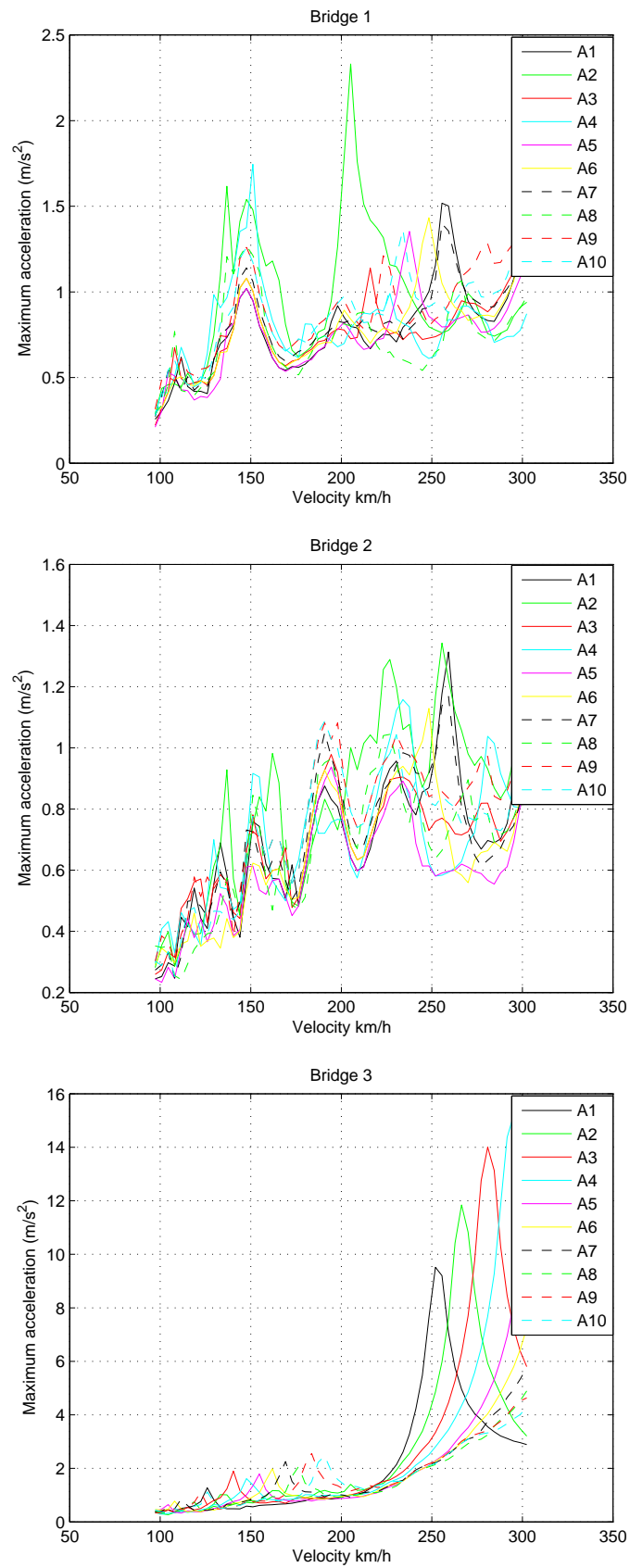
Figure 4.13: *The maximum acceleration as a function of the velocity of the loads for bridge 1,2 and 3.*

4.4. Discussion

Modelling a bridge using a large number of elements, short time-steps and with all eigenmodes produce results with great accuracy but the computer-time needed to perform the analysis are extensive. In order to make use of the design tool the computer-time needs to be reduced but the accuracy had to be kept. This fact had to be considered when choosing an appropriate number of elements, value of time-step and number of eigenmodes included as standard settings in the reduced bridge model.

The general setups for the program had also got to be adopted to the positions of the supports of the bridge. This was due to the fact that the support had to be placed in a predefined element node. More elements in the model made it possible to have an increased accuracy of the position of the supports, this also contributed to a better accuracy of the results.

By using more time-steps per element the accuracy of the result was increased but the computer-time needed to perform the analysis was clearly extended. The result indicates that it is better to increase the number of elements instead of increasing the number of time-steps per element.

The dynamic response of the three bridges can be described in a good way by including the first four eigenmodes in the analysis. According to the design codes, only the first eigenmode for bridge 1 and 2 has to be included in the analysis, but then the maximum acceleration is underestimated. The computer-time needed to preform an analysis was negligible when comparing 1 and 4 included eigenmodes. Therefore, the four first eigenmode is recommended to be included in these analyzes.

5. Other methods of modelling loads

There are different approaches of modelling the train loads compared to what the design codes are specifying. In this chapter two other approaches will be described and analyzed.

5.1. Distributed load model

When performing a static analysis of railway bridges with sleepers, it is, according to the Swedish design code[1], possible to model the load from the train as a distributed load. This is based on the theory that a point load on the rail is assumed to be distributed by the sleepers and by the ballast layer, before affecting the load-bearing structure. This phenomena is illustrated in figure 5.1. In this section a dynamic model that prescribes a distributed load will be described.

5.1.1. Model
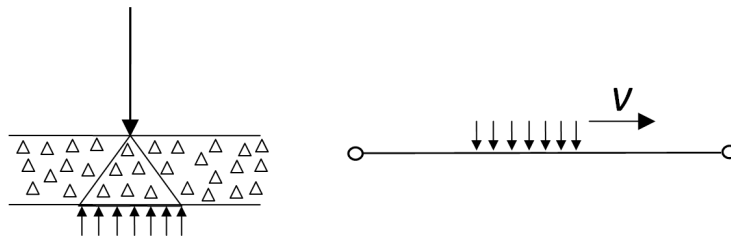
Figure 5.1: *The distributed load model.*

The distributed load was modelled using point loads equally placed over a pre-defined area. The size of the initial load was equal to the sum of all the modelled point loads. The model is illustrated in figure 5.1.

The load matrix $\mathbf{F}$, was developed as a MATLAB function. The program code is a modification of the code developed in chapter 4. The difference was that the load of a train axle was modelled as an adjustable number of point loads equally placed over an adjustable length on the bridge. Apart from that, the same methods as described in chapter 4 was utilized. The code have comments and can easily be modified. The commented code is found in appendix B.

41

5.1.2. Results and analysis

In order to illustrate the importance of modelling a distributed load using a sufficient number of equally placed point loads, the maximum accelerations produced by train set A1 in a bridge was studied. The bridge had a length of 10 meter, I was equal to 1 m^4, Young's module was equal to 40.8 GPa and the maximum accelerations were produced by train A1. Figure 5.2 shows the acceleration versus velocity with ten plots corresponding to the number of equally placed point loads that were used when modelling the load of a train axle.

The results showed that the accelerations had a slight increase when a train axle was modelled using more point loads. The increase of the maximum acceleration was levelled off when the train axle was modelled by approximately 100 equally spaced point loads.

In this case the difference of the maximum acceleration was approximately 4 percent when comparing with the result of using 3 and 102 point loads.

In order to estimate the effects a distributed load had on the maximum accelerations when the stiffness of a bridge was varied, the same bridge but using three different values of the Young's module was again studied. The results were compared with the the results obtained when modelling the load of a train axle as a single point load. The dimension of the bridges and the results are shown in figure 5.3. The forces from the axles of train A1 was modelled as 100 point loads for each axle.

The results showed that a distributed load model generated less dynamic response compared with the point load model. The difference was more obvious in bridge 3, especially at the resonance peaks. In bridge 1 the difference of the maximum acceleration was 1.8 m/s^2, in bridge 2 0.6 m/s^2 and in bridge 3 0.14 m/s^2. The maximum accelerations of a bridge were increased when a bridge was less stiff, in this case had a lower Young's module. The results does imply that the influence of distributed loading increases when the maximum accelerations increases. Standard specifies that the maximum acceleration must be below 3.5 m/s^2, the bridge with the largest value of Young's module was the only bridge that fulfilled that requirement. The bridge had also the smallest impact on the results.
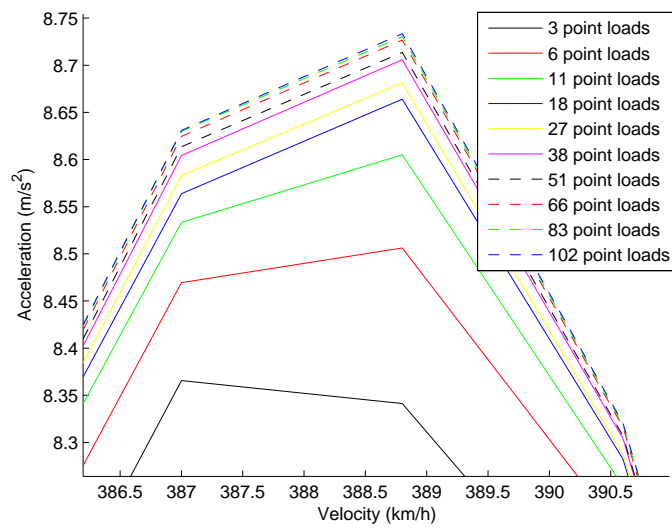
Figure 5.2: *The maximum acceleration as a function of the velocity using a different number of point loads to model the distributed load.*
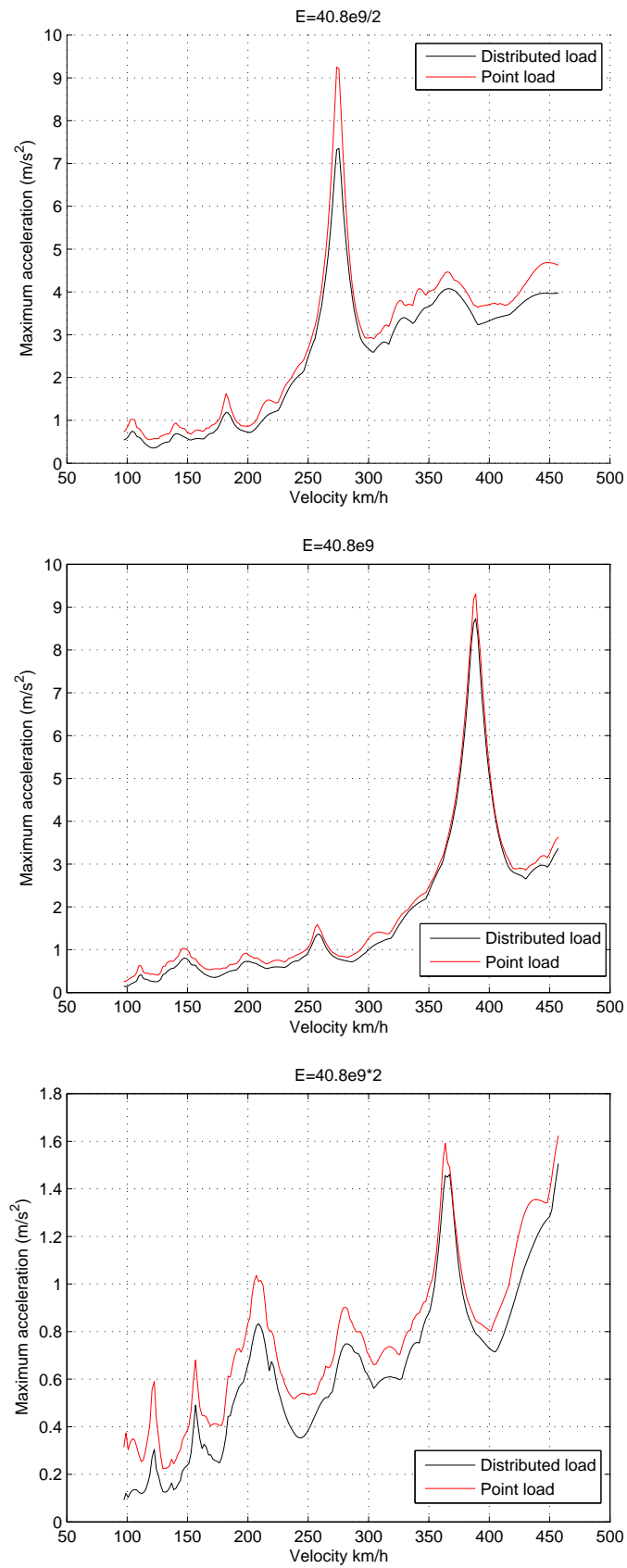
Figure 5.3: *The maximum acceleration as a function of the velocity using a different E-modules.*

5.1.3. Discussion

The results of the analysis shows that a smaller dynamic response is obtained when the train axel is modelled as a distributed load compared with a point load. The varied impact on the results implies that more bridges needs to be estimated before conclusion of an expected reduction of the response can be made.

In this work, only the influence of a distributed load on the vertical eigenmodes was studied. However, a distributed load may have an influence on the excitation of transversal eigenmodes since the load also must be equally spread out in the transversal direction. This may increase the vertical acceleration of the bridge and must also be further studied in order to estimated the total impact of a distributed load.

5.2. Interaction load model

An other method of modelling the load of the train is called the interaction load model. The train is then modelled using a mass that corresponds to the mass of a proportional part of the train connected to a spring that corresponds to the stiffness of the train. The train model can be more advanced and realistic, but the aim of this work was just to examine the fundamental differences in modelling the load by a spring and a mass as compared with the other used methods.

Forces acting on the bridge when using the interaction load model depends on the weight of the modelled train, the motion of the bridge and the motion of the train masses. I.e. the forces will depend on the interaction between the train and the bridge. As a result, the loads will become varying with time and not constant as in earlier described load models. An illustration of the interaction load model is shown in figure 5.4
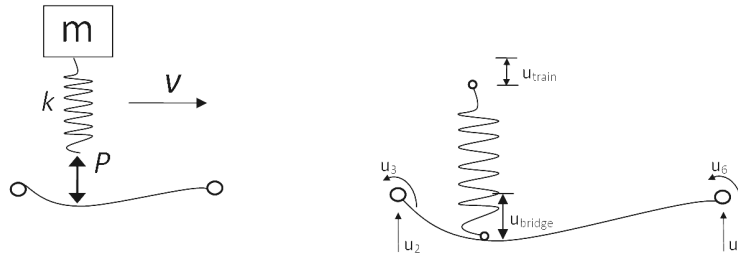
Figure 5.4: *The load model of interaction between train and rail.*

5.2.1. Model

The model was developed as a MATLAB function called *interactionprogram*. The set-up of **K**, **C** and **M** was the same as in the earlier described bridge models. The **F** vector had to be recalculated for every time step due to the fact that the value of the load of the train model differed. The procedure of how to construct this load vector is, with the use of figure 5.4, described in six steps below:

1. The train model moves a distance Δx. The value of the force P acting between the bridge and the load was calculated in the previous time-step.

2. The equivalent node forces is calculated using equations (4.2) - (4.5) and the load vector **F** can be established.

3. By solving the equations of motion (2.7) for this time-step the displacement in the element nodal points are calculated.

4. With the use of the shape functions of a beam the displacement of the train model can be calculated according to:

$$N_2^e = 1 - 3\frac{x^2}{L^2} + 2\frac{x^3}{L^3} \tag{5.1}$$

$$N_3^e = x(1 - 2\frac{x}{L} + \frac{x^2}{L^2}) \tag{5.2}$$

$$N_5^e = \frac{x^2}{L^2}(3 - \frac{x}{L}) \tag{5.3}$$

$$N_6^e = \frac{x^2}{L^2}(\frac{x}{L} - 1) \tag{5.4}$$

$$u_{bridge} = N_2^e \cdot u_2 + N_3^e \cdot u_3 + N_5^e \cdot u_5 + N_6^e \cdot u_6 \tag{5.5}$$

5. The deflection u_{bridge} is used as a boundary condition in the train model. By solving equation of motion u_{train} is calculated.

6. The force P used in next time step is calculated according to:

$$P = k(u_{train} - u_{bridge}) \tag{5.6}$$

5.2.2. Result and analysis

While developing the program it became clear that the size of a time-step needed to be much shorter in order to provide accurate results as compared to the other used methods. The reason for this was that a small out of balance error appear when constructing the load vector. In this work, a smaller time-step was employed instead to produce a smaller error.

Figure 5.5 shows the response of the interaction load using a too large time-step. The result shows that the solution become clearly unstable.

In order to illustrate the difference of using the interaction load and constant point load, a 10 meter simple supported bridge was studied. The varied load consisted of a single mass, m, with a suspension, k, and the value of the constant load was equal to m·g. The value of m was based on the point load value of the HSLM load, described earlier in chapter 3, and the value of k is assumed to correspond to a 10mm deflection by the weight of the train.

Figure 5.6 shows the dynamic response obtained by analyzing a bridge using the same bridge properties as the bridge presented in figure 5.5, but a smaller time-step was used. The result was compared with the dynamic response obtained by modelling the load of a train axle using a point load.

The result shows that the responses of both loads were clearly different. The maximum displacement using an interaction load was larger and the maximum peak of the displacement was shifted more to the right side of the bridge. The reason for this is that the force of the mass increases when the deflection of the bridge starts to decrease. This could be explained by the fact that a force is needed to change the motion of a mass.

The result indicated that there was a different dynamic response in a bridge depending on the used method of modelling a train axle.

An additional factor that may had an influence on the result was the fact that the train model did not include a damper. An more precise model of a train would have had a damper, and that would have had reduced the dynamic response. This fact may also be the explanation to some of the difference that appeared when comparing maximum accelerations in figure 5.5.

The fact that the maximum displacement appeared in a different position in the bridge can not be explained due to potential wrong mass and that no damping had been used in the train model. This is an effect of the motion of the mass, that a constant load model, not considers. This implies that other modes of vibrations may be excited and the dynamic response may therefore be different then what is expected.
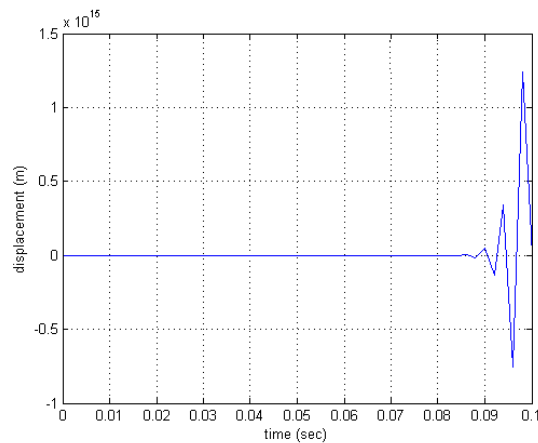
Figure 5.5: *The response obtained when a too large time step has been used.*
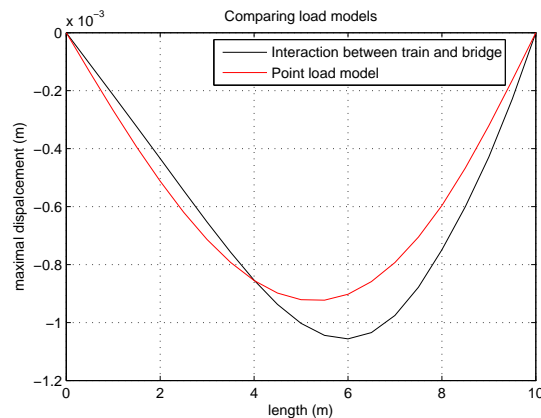
Figure 5.6: *Comparing the maximum displacement obtained when using a point load and when considering the interaction between train and bridge.*

5.2.3. Discussion

In order to make a more accurate analysis of the effect that the interaction between train and rail had on the dynamic response the program must be further developed. That would demand an increase of the computer effort, and hence more time to perform the analysis. Since the aim of this work was to develop a simplified tool, that could perform a quick analysis, this was out of the scope of this work.

It was obvious that the interaction between train and rail had an effect on the dynamic response and had to be considered in order to perform an absolutely accurate dynamic analysis. The question may be asked, how important it was to improve only this detail of the analysis. It was possible that the dynamic response was to be more accurate if another part of the model was modelled more in detail. For instance the eigenmodes of the model could be recalculated at every time-step in order to include the mass, damping and stiffness of the actual train in the bridge model. However, the Swedish design codes prescribes that the effects from interaction between train and rail may have to be checked for bridges with a span longer then 30 meters. And this fact implies that the studied effect may have an important impact.

6. Summary of modelling

the choice of modelling the load was set to the point load model and

In order to make the developed design tool easy to use and to perform analysis quick and accurate, the numbers of input parameters were minimized. This was made by setting some of the properties of the reduced model as fixed standard values. The values were based on the result of the modelling studies presented in chapter 5.

The following properties were set to fixed values;

Initial load method = Preloaded
Number of element = 4 × the length of the bridge
Time step per element = 10
Precision of the boundary condition = .25 meter
Number of included eigenmodes = 4

The only variable properties that has to be set by the user are:

- The length of the bridge
- Positions of the supports
- The damping value
- Young's module
- Moment of inertia
- The mass per meter

6.1. Verifying the presented calculation tool

In this section dynamic analyzes will be presented for two different bridges with realistic dimensions and materials. The analyzes were performed using both a conventional finite element program called ADINA as well as the developed calculation tool. The results from the two models will be compared and analyzed.

6.1.1. Bridge model 1

The first bridge that was analyzed was a five span railway bridge with a track in the center of the bridge. A side view of the bridge and a view of the cross section is found in figures 6.1 and 6.2.

The choice of analyzing a bridge with these properties was to verify that the simplified model could provide an accurate result on a railway bridge with a wide track and several spans.
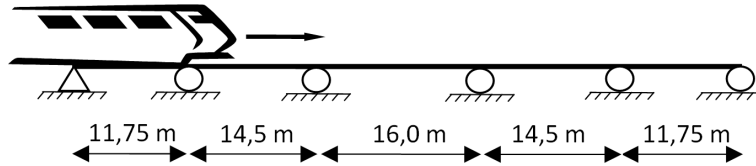
Figure 6.1: *A side view of bridge model 1.*
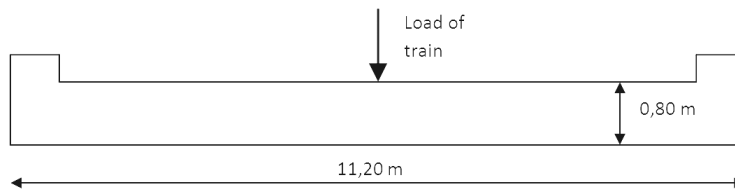
Figure 6.2: *A view of the cross section of bridge model 1.*

Model and input data

The 3-dimensional ADINA model of the bridge was modelled using beam elements with the assumptions of a lumped mass, which means that the mass representing each element is located in the element nodes. The supports of the model were modelled as line boundary conditions. The mesh of the beam elements and the extension of the boundary conditions can be found in appendix D.

Masses lumped in supports, where the deflection were set to zero, were not active during the dynamic analysis. Due to this fact and to the geometry of the bridge mesh, less mass were active in the 3-dimensional model compared to the simplified model that is modelled using a constant mass. To be able to compare both models correctly, the active masses must be equal. The mass per meter that was to be used in the reduced model, had therefore to be calculated to represent the mass of the three-dimensional model. An approximate calculation of the reduction of the mass is presented below:

$$\chi = \frac{\triangle BC}{\triangle Nodes} \tag{6.1}$$

χ = The reduction of mass
$\triangle BC$ = Number of node points in a BC
$\triangle Nodes$ = Total number of node points

The following input data was used to model bridge 1:

Length of the bridge = 68.5 meter
Positions of the supports = [0 11.75 26.25 42.25 56.75 68.5]
The damping value = 1.5 percent
Young's module = 40.8 GPa
Moment of inertia = 0.4632 m^4
Mass per meter = 32523 kg/m (The reduction of mass = 6 percent)

The maximum accelerations of the bridge are calculated in steps of 5 km/h, for the ADINA model, and in steps of 3.6 km/h for the reduced model.

Results and analysis

The shape of the eigenmodes and the corresponding eigenfrequencies from the two models are shown in figure 6.3 and 6.4 respectively. These results shows that both the shape and the frequency of the first eigenmode were very similar for the ADINA and the reduced model. When comparing the shapes of the second and third eigenmodes they were also similar, but the eigenfrequencies were clearly differing. The second eigenmode differ by 21 percent and the third eigenmodes with 40 percent. The fourth eigenmode of the simplified model corresponds to the seventh eigenmode of the ADINA model. These mode shapes were similar, but the difference of the frequency was equal to 254 percent. Eigenmode number four, five and eight as shown in figure 6.4 were torsional eigenmodes and was not calculated by the two-dimensional reduced model. Eigenmode six was a vertical mode and there was no corresponding mode in the four first eigenmodes in figure 6.3.

This result proves in this case that the calculated shapes of the eigenmodes were good approximations, but the match between the eigenfrequency was reduced for each eigenmode.

In figure 6.5, a comparison of the acceleration versus velocity spectra for train models A1, A2 and A10 are shown. The comparison clearly show that the simplified model provide results, for this type of bridge, that are close to the results provided by the more advanced model. The acceleration values are in general more accurate for speeds exceeding 200 km/h. For speed below 200 km/h, the results are still of the same magnitude, but less accurate.

In order to illustrate the influence different eigenmodes have on the results, an analysis was performed where the number of included eigenmodes were varied. The result of the analysis can be found in figure 6.6 which shows the influence of the first four eigenmodes on the total response.

The difference found for the maximum acceleration, as shown in figure 6.5, may to some extent be explained by the fact that from the ADINA model, the acceleration was calculated in time-steps of 5 km/h, which may not have been accurate enough. Some of the variation may also be due to the fact that the calculated frequencies of the eigenmodes was not as accurate as the first ones. The frequencies calculated using the simplified model had a tendency to be higher than the ADINA model. Therefore, according to the design codes described in chapter 3, more eigenmodes should have been included in the analysis when using the ADINA model.
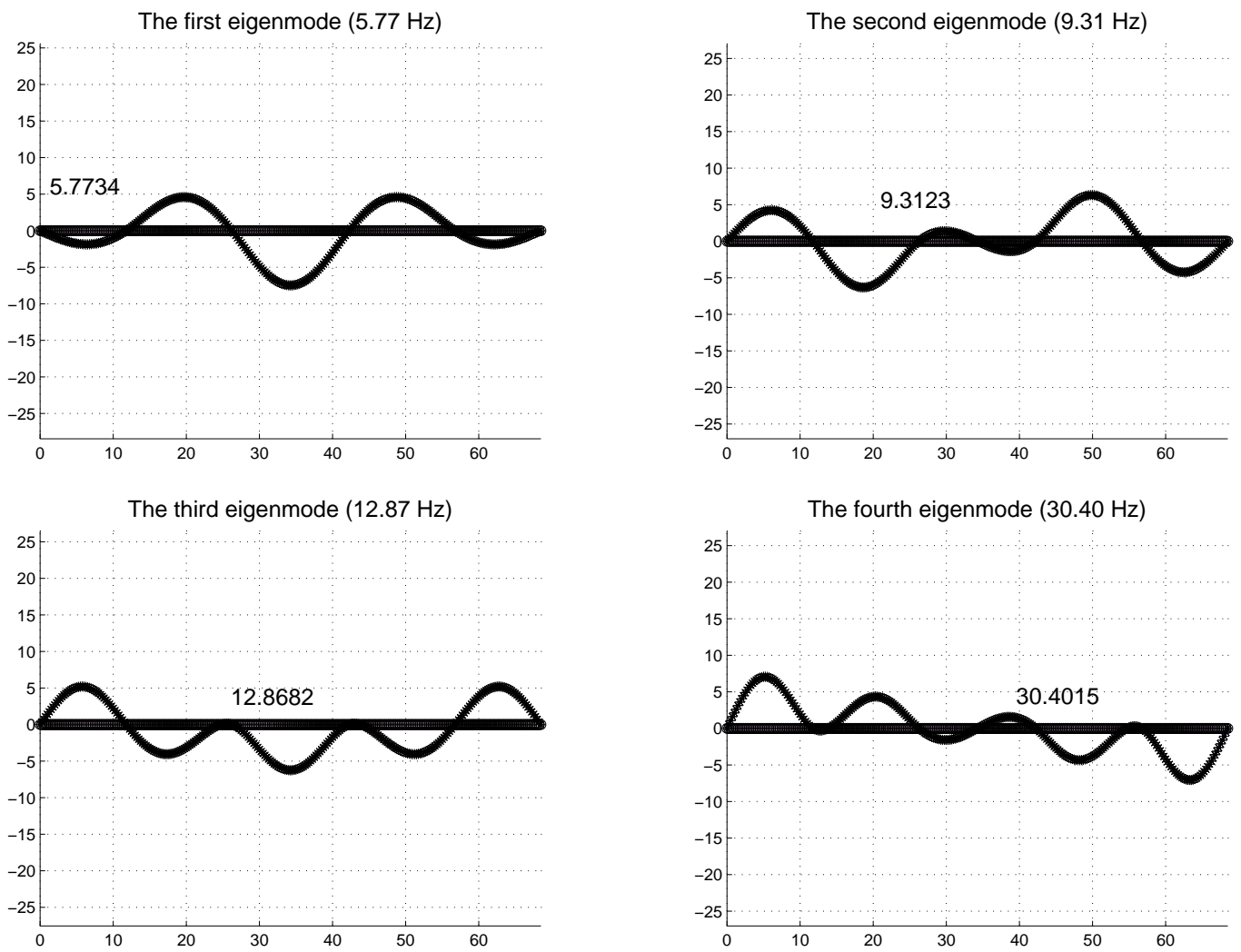
Figure 6.3: *The first four eigenmodes calculated using the reduced model.*

Figure 6.4: *The first eight eigenmodes calculated using ADINA.*

Figure 6.5: *A comparison of the maximum acceleration as a function of the velocity obtained by the ADINA program and the simplified calculation tool.*
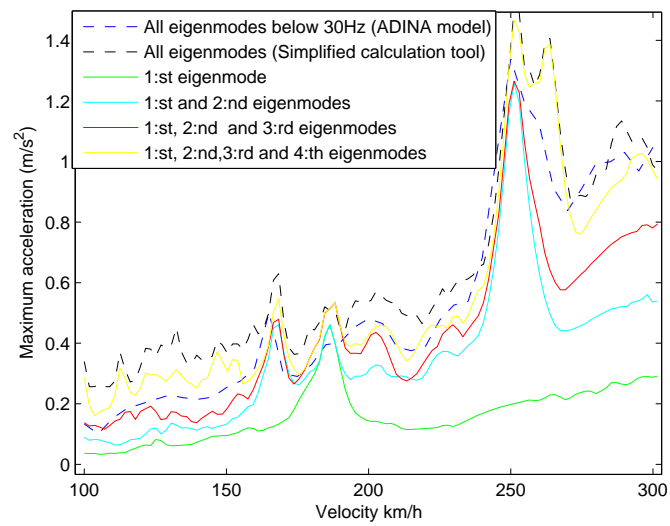
Figure 6.6: *The maximum acceleration as a function of the velocity calculated including a varied number of eigenmodes.*

6.1.2. Bridge model 2
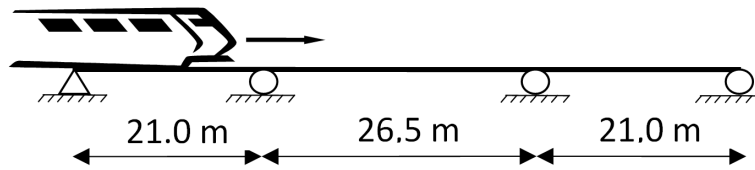
Figure 6.7: *A side view of bridge model 2.*

The second bridge to be analyzed with the two models was a three span railway bridge with centric load. A side view of the bridge is found in figures 6.7. The choice of analyzing this type of bridge was to study what effect the number of supports had on the results. The same material, total bridge length, properties and cross-section geometry as for Bridge model 1 was assumed.

Model

The 3-dimensional ADINA model was modelled using beam elements with a lumped mass. The mesh was the same as in the previous analyzed bridge, but the boundary conditions were changed.

Due to the fact, that the boundary conditions were changed, a new value for the active mass had to be calculated. This was performed using equation (6.1).

The following input data was used to model bridge 2:

Length of the bridge = 68.5 meter
Positions of the supports = [0 21.0 47.5 68.5]
The damping value = 1.5 percent
Young's module = 40.8 GPa
Moment of inertia = 0.4632 m^4
Mass per meter = 32866 kg/m (The reduction of mass = 4 percent)

The maximum accelerations of the bridge are calculated in steps of 5 km/h, for the ADINA model, and in steps of 3.6 km/h for the reduced model.

Results and analysis

A comparison between the shape of the eigenmodes and the corresponding frequencies, calculated with both methods, are shown in figure 6.8 and 6.9. The accuracy of the results have the same tendency as the results from the analyzes of Bridge model 1, the shapes of the eigenmodes were similar, but the match between the eigenfrequency was reduced for each eigenmode. The difference in the eigenfrequencies for
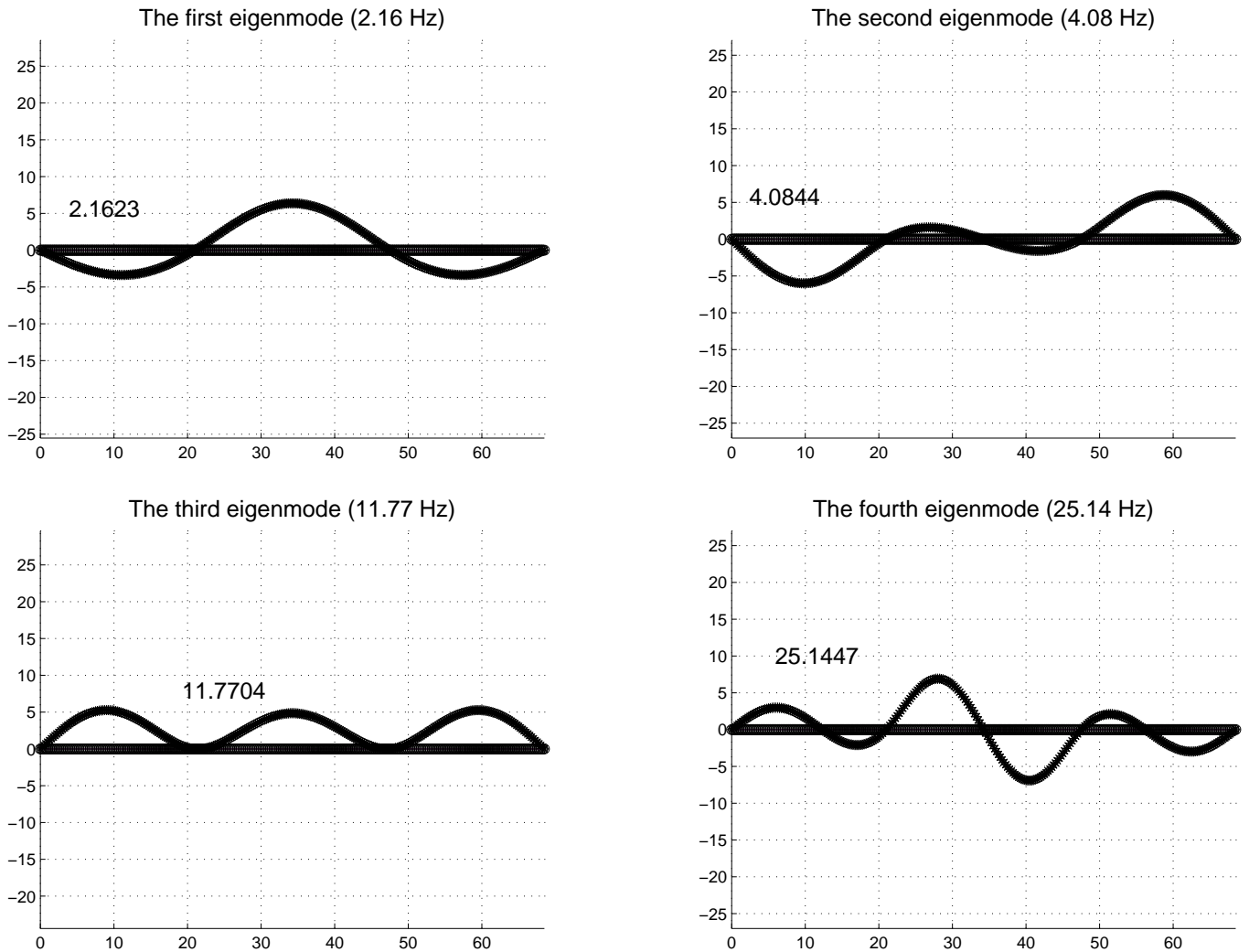
Figure 6.8: *The first four eigenmodes calculated using the simplified tool.*

the four first modes were calculated with bridge model 1, compared with the result of bridge model 2. The results showed that the accuracy of bridge model 2 is better.

The result of the dynamic analysis, using the reduced model and the ADINA model, were compared and presented as acceleration/velocity spectrums for train A1, A2 and A10. This is shown in figure 6.10. The results show that the reduced model provide accurate results also when modelling Bridge model 2. Comparing the results in figure 6.5 and 6.10 show that the accuracy is better for bridge model 2.

The differences of the results may also in this case be depending on the fact that the results from both models were calculated using different time-steps. Some of the difference may also depend on that the models not have the exactly same corresponding eigenfrequencies.
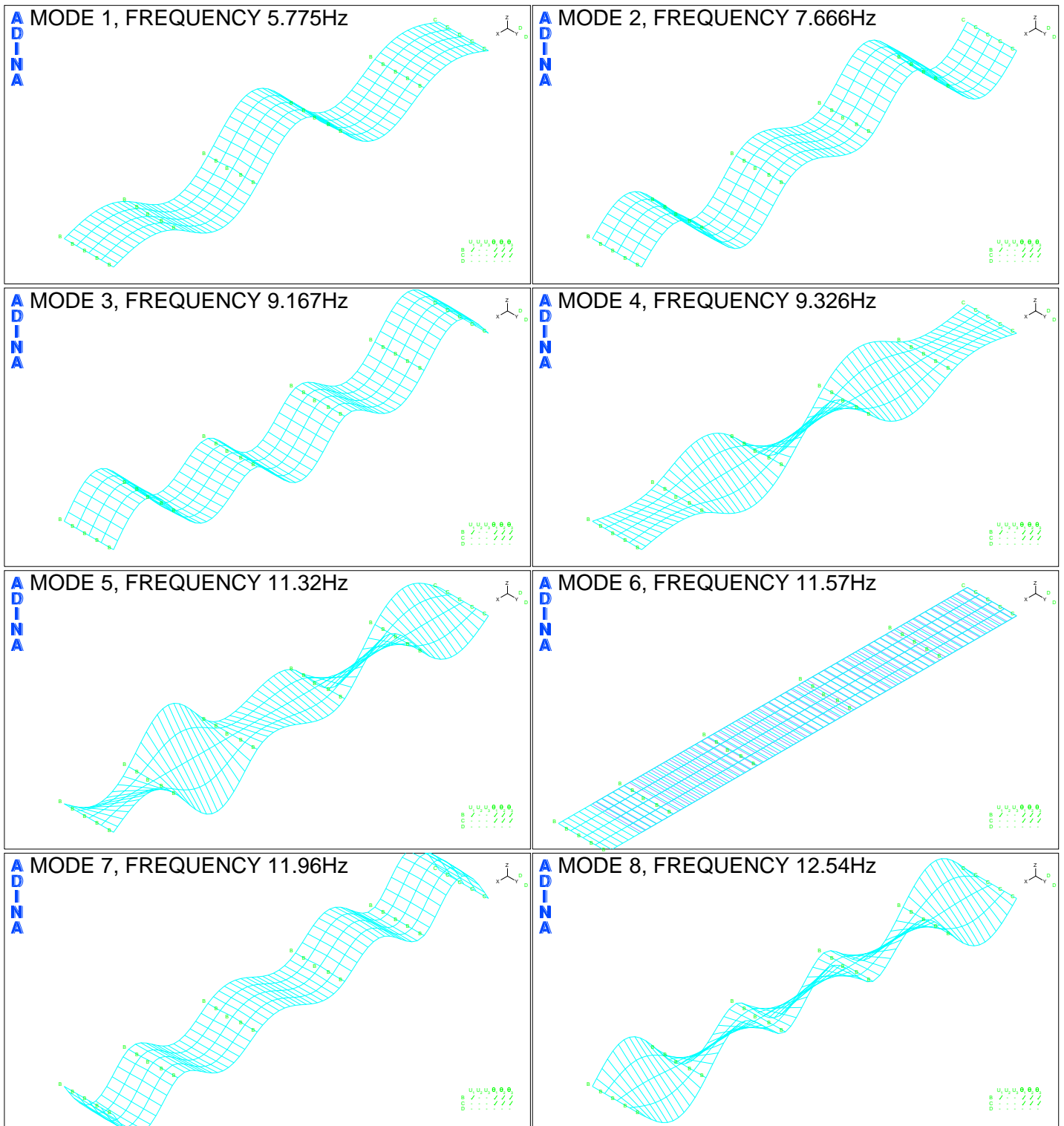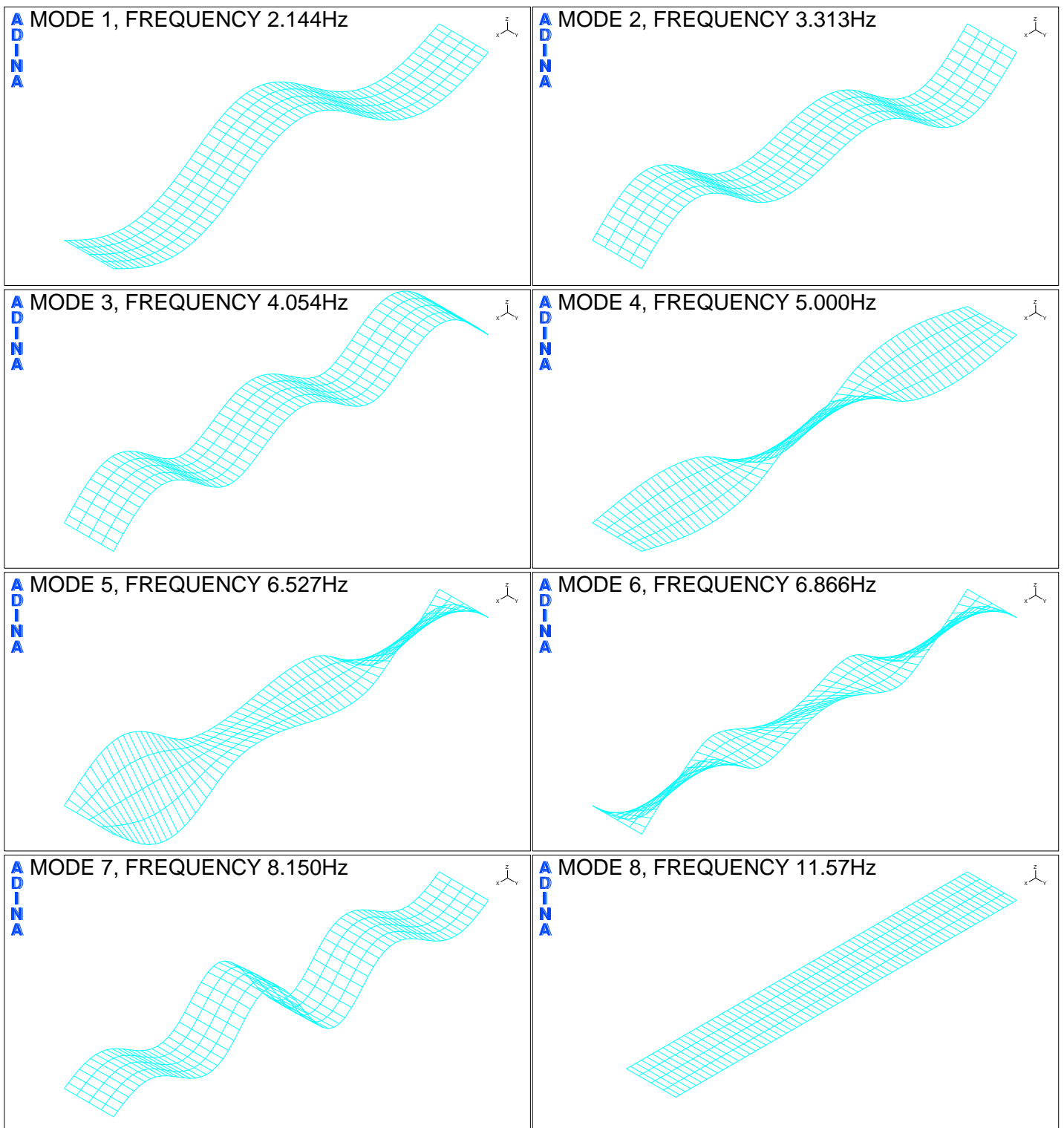
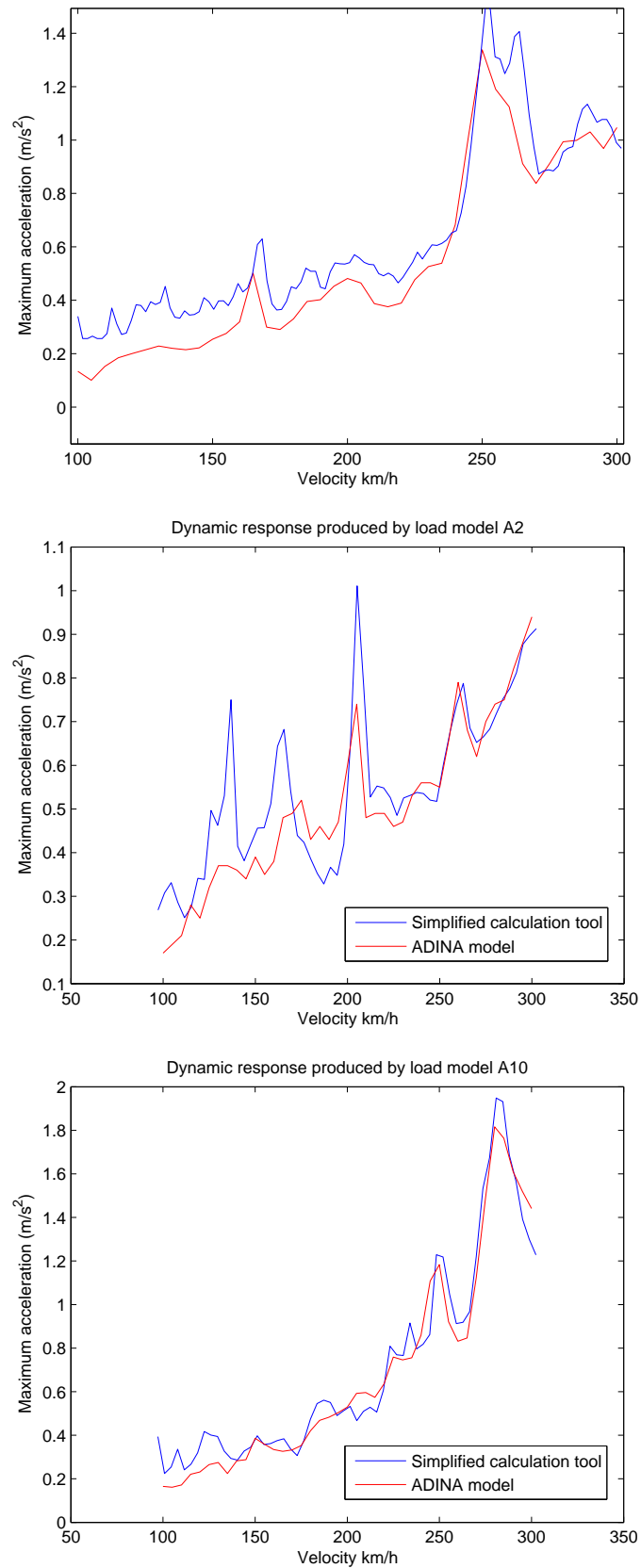Figure 6.9: *The first eight eigenmodes calculated using ADINA.*
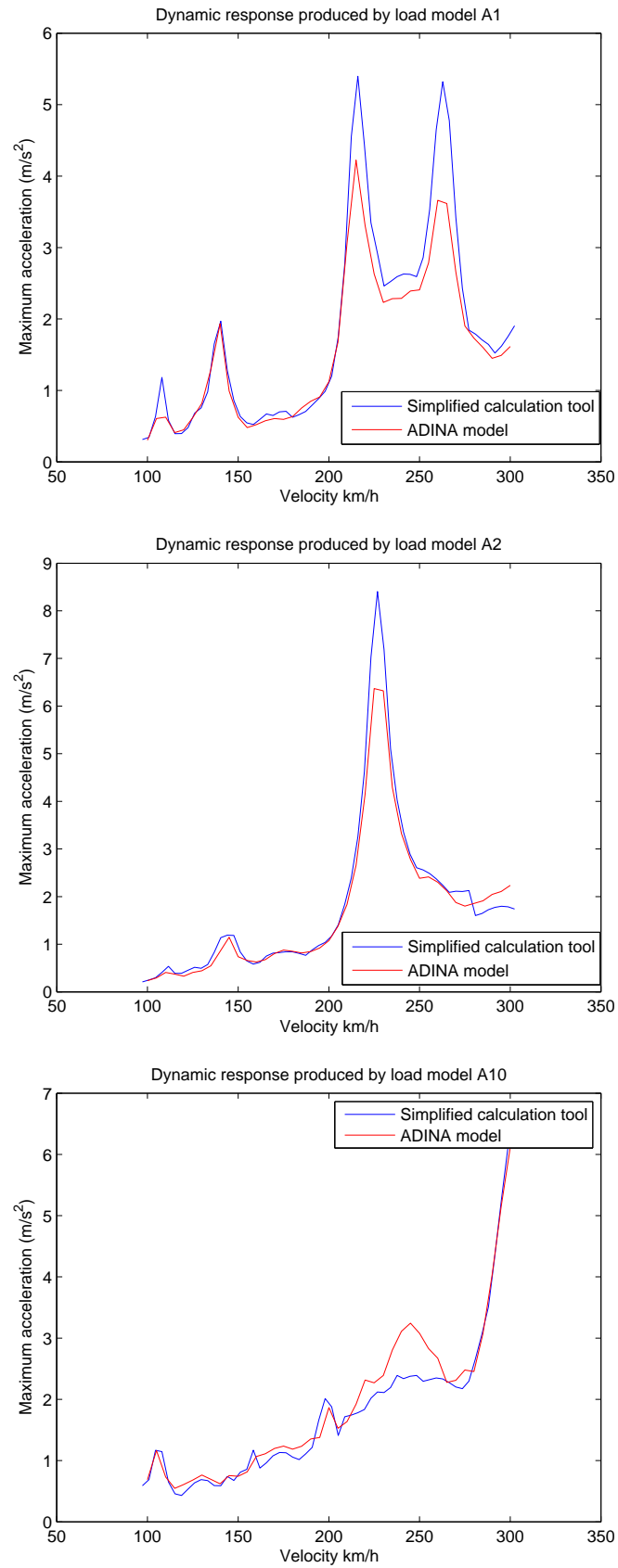
Figure 6.10: *A comparison of the maximum acceleration as a function of the velocity obtained by the ADINA program and the simplified calculation tool.*

7. CONCLUSIONS

7.1. Summary of findings

In this work a reduced model for dynamic analysis of railway bridges was developed. Results of this study have proved that a reduced model can provide basically the same results as a more advanced model. The time needed to model the bridge and to perform the analysis was approximately 10 minutes when using a standard home pc, which was a significant reduction of the time needed to make an advanced model and to perform the analysis.

Modelling a bridge using a large number of elements, short time-steps and with all eigenmodes have proved to produced results with great accuracy but the computer-time needed to perform the analysis was extensive.

Results of this study indicates that it is better to increase the number of elements instead of increasing the number of time-steps per element. The results also showed that the dynamic response can be described in a good way by including the first four eigenmodes in the analysis.

A smaller dynamic response was obtained when the train axle was modelled as a distributed load as compared with a point load. The results implies that more bridges needs to be estimated before conclusion of an expected reduction of the response can be made.

It was obvious that the interaction between train and rail had an effect on the dynamic response and had to be considered in order to perform an absolutely accurate dynamic analysis. This type of analyis, however, increases the effort and the time needed to perform the analysis, and was not in the scope of this work.

The reduced model was only verified by two more advance models and the results showed that the accuracy of the results differed. In order to estimate the limitation of the model, more bridges must be analyzed and verified. The best way of verifying a model would have been to compare the results using data measured on an actual bridge or to compare the result using an extensively advanced and already verified model. By doing this the reduced model can be optimized.

Nevertheless, the results of this study shows that the reduced model may be very

useful early in a project. The program may also be used to perform other then dynamic calculations. For instance the maximum deflection for a static load can be calculated by having train A1-A10 pass the bridge with a low velocity (e.g. 0.01 m/s). The deflection will then be calculated for all the load cases a train may possibly generate and this way the maximum deflection can be found.

7.2. Proposal for further work

The reduced model must be verified further. More bridges with a varied geometry, number of supports and material properties must be analyzed and compared with results of more advance models. A good way of comparing the results of the reduced model would be to use data measured on an actual bridge or to compare the result using an extensively advanced and already verified model.

The effects of an eccentric load must also be estimated in order to find how large impact the negligible of the transversal modes have on the total dynamic response.

The MATLAB function and the reduced model can also be further developed and for instance include:

- An improved interface.
- The effects of transversal eigenmodes by using 3-dimensional beam elements.
- Allow different stiffness and masses in each beam element.
- Develop the software so it is possible to add more masses in element nodes.
- Calculate section forces in each element.
- Analyze the effect from calculating new eigenmodes at every time-step, when the masses of the train is included in the bridge model.

BIBLIOGRAPHY

[1] Banverket, (2004), *BV BRO Banverkets ändringar och tillägg till Vägverkets Bro, 2004*, Utgȧva 7. BVS 583.10

[2] ERRI D214 committee, (1999), *Rail bridges for speed > 200 km/h*; Final report, Part A, Synthesis of the result of D 214 research, Part B, Proposed UIC Leaflet, European Rail Research Institute (ERRI).

[3] Ray W. Clough and Joseph Penzien, (1982), *Dynamics of Structures*, McGraw-Hill, New Jersey, USA.

[4] Timoshenko, S. and Young, D., (1955) *Vibration problems in engineering*, Van Nostrand, 3rd ed.

[5] Fryba, L., (1996), *Dynamics of railway bridges*, Thomas Telford, Great Britain, London

[6] Gabaldón Felipe, J.M. Domnguez, J.A. Navarro, F. Riquelme, (2002) *Dynamic analysis of hyperestatic structures under high speed train loads*, Technical Report, Spanish, Madrid.

[7] Björn Lundin, Philip Mȧrtensson, (2006), *Finding general guidelines for choosing appropriate cut-off frequencies for modal analyses of railway bridges trafficked by high-speed trains*, Division of Solid Mechanics, Lund University, Sweden, Lund.

[8] Ottosen N-S., Pettersson H., (1992), *Introduction to the Finite Element Method*, Prentice Hall Europe, Great Britain.

[9] Oscar Jaramillo de Leon, Kasper Lasn, (2008), *Dynamcs of railway bridges subjected to high-speed trains - SDOF models for the approximate evaluation of resonance*, Chalmers University of Technology, Gteborg, Sweden.

[10] Ekström D, Kieri L.-L,(2007), *Dynamic analysis of railway bridges*, Chalmers University of Technology, Gteborg, Sweden.

[11] Austrell, P-E., et. al., (2004), *CALFEM A finite element toolbox version 3.4.* KFS i Lund AB, Lund.

[12] ADINA, (2004), *Theory and Modelling Guide Version 8.2.2*, ADINA R&D, Inc., Volume 1, Watertown, USA.

[13] Heyden, S., et. al., (2005), *Introduktion till strukturmekaniken*, KFS Lund AB, Lund, Sweden.

[14] Wiberg Johan, (2006), *Bridge Monitoring to Allow for Reliable Dynamic FE Modelling*, Royal Institute of technology, Stockholm, Sweden.

Appendix

A. MATLAB code - The reduced model

```
%-----------------------------------------------------------------------
% PURPOSE:
% Compute the dynamic response in a high-speed railway bridge.
% The load of the train are modelled as point loads.
%-----------------------------------------------------------------------
% INDATA:
% L_bridge = Length of the bridge
% Bridge_support = Position of the supports
% damping = The damping value of the bridge
% E = Young's modoulous
% I = Moment of inertia
% m = The mass per meter
%-----------------------------------------------------------------------
% AUTHOR: Andreas Gustafsson 2008-12-10
%
% Copyright (c) Andreas Gustafsson
%-----------------------------------------------------------------------

clear all
close all
format long
%--------------------------- Indata ------------------------------------

L_bridge =10;
Bridge_support = [0 L_bridge];
damping = 0.015;


% -------- Material data ---------------

E = 40.8e+009;
I = 0.4632;
m = 34235*0.96;


%-------- Fixed parameters ---------------------------------------------
A = 80.7875;
time_step_element = 10;
number_of_element = L_bridge*4;
number_of_eigenvectors = [1:4];
load_on_steps = 2;
```

```matlab
%-------- Calculated parameters -------------------------------------------
dx = L_bridge/(time_step_element*number_of_element);
dgf = (number_of_element+1)*3;
L_element = L_bridge/number_of_element;
bc = [round(Bridge_support/L_element)*3+2 round(Bridge_support(1)/L_element)*3+1];
% --------------------- Topology matrix -----------------------------------
disp('Calculating element geometry')
[Ex,Ey,Edof]=geometry2(number_of_element,L_bridge);
disp('Done')
%-------- Calculating damping parameters and eigenmodes --------------------
disp('Calculating eigenmodes')
[omega1,omega2]=eigenfrequency(number_of_element, L_bridge, bc,
damping, E, A, I, m, Ex, Ey, Edof)
disp('Done')
omega2 = 1.884955592153876e+002;  % temporary value
a0 = damping*2*omega1*omega2/(omega1+omega2);
a1 = damping*2/(omega1+omega2);
ep = [E A I m [a0 a1]];
%-------- Stiffness, mass, force and damping matrix bridge -----------------
disp('Assemble stiffness-, mass and damping matrix')
K = zeros(dgf);
M = zeros(dgf);
C = zeros(dgf);
for i=1:number_of_element
    [Ke,Me,Ce] = beam2d(Ex(i,:),Ey(i,:),ep);
    K = assem(Edof(i,:),K,Ke);
    M = assem(Edof(i,:),M,Me);
    C = assem(Edof(i,:),C,Ce);
end
disp('Done')
% --------------------- Eigenvalue analysis -------------------------------
disp('Calculating eigenvalue')
[La,Egv] = eigen(K,M,bc);
disp('Done')
% ----- Reduced system matrices -------------------------------------------
kr = sparse(diag(diag(Egv(:,number_of_eigenvectors)'*K*
Egv(:,number_of_eigenvectors))));
mr = sparse(diag(diag(Egv(:,number_of_eigenvectors)'*M*
Egv(:,number_of_eigenvectors))));
cr = sparse(diag(diag(Egv(:,number_of_eigenvectors)'*C*
Egv(:,number_of_eigenvectors))));
% ----- Initial condition -------------------------------------------------
dr0=zeros(length(number_of_eigenvectors),1);
vr0=zeros(length(number_of_eigenvectors),1);
% ----- Output parameters -------------------------------------------------
ntimes=[0.1:0.1:1];
nhistr=number_of_eigenvectors;
```

```
nhist=[2];
for i=1:number_of_element
    nhist = [[nhist] i*3+2];
end
%---------------------- Start loop -------------------------------------------
for Load_model = 1:10
% ---------------------- Train model -----------------------------------------
    disp('Reading train model')
    [HSLM_length,F_train]=hslm(Load_model);
    disp('Done')
    for speed_increase=0:50
        %-------- Calculated parameters -----------------------------------------
        speed = 27.778 + speed_increase*0.5;
        dt = L_bridge/(speed*time_step_element*number_of_element);
        time_passing_bridge = L_bridge/speed;
        HSLM_time = HSLM_length/speed;
        total_time = time_passing_bridge + max(HSLM_time) + load_on_steps*dt;
        % ------- Force matrix -----------------------------------------------
        f = zeros(dgf,(total_time/dt + 1));
        [t,g] = gfunc([0 0;load_on_steps*dt -1],dt);
        for k=1:length(HSLM_time)
            time_hist_HSLM = ceil(HSLM_time(k)/dt);
            if time_hist_HSLM == 0
                time_hist_HSLM = 1;
            end
            f(2,time_hist_HSLM : load_on_steps+time_hist_HSLM-1) =
            g(1:length(g)-1) + f(2,time_hist_HSLM :
            load_on_steps+time_hist_HSLM-1);
            for j=1:number_of_element
                global_u2 = 3*j-1;
                global_u3 = 3*j;
                global_u5 = 3*j+2;
                global_u6 = 3*j+3;
                time_hist_element = (j-1)*time_step_element;
                for i = 0:round(L_element/dx)-1
                    %------------ Calculating parameters --------------------
                    a = dx*i;
                    b = L_element-dx*i;
                    %------------ Calculating F matrix ----------------------
                    f( global_u2 ,(time_hist_HSLM + time_hist_element + i +
                    load_on_steps)) = F_train*b^2/L_element^2*(1+2*a/L_element)
                    + f( global_u2 ,(time_hist_HSLM + time_hist_element + i +
                    load_on_steps));
                    f( global_u3 ,(time_hist_HSLM + time_hist_element + i +
                    load_on_steps)) = F_train*a*b^2/L_element^2 + f( global_u3
                    ,(time_hist_HSLM + time_hist_element + i + load_on_steps));
                    f( global_u5 ,(time_hist_HSLM + time_hist_element + i +
```

```
                  load_on_steps)) = F_train*a^2/L_element^2*(1+2*b/L_element)
                  + f( global_u5 ,(time_hist_HSLM + time_hist_element + i +
                  load_on_steps));
                  f( global_u6 ,(time_hist_HSLM + time_hist_element + i +
                  load_on_steps)) = -F_train*a^2*b/L_element^2 + f( global_u6
                  ,(time_hist_HSLM + time_hist_element + i + load_on_steps));
              end
          end
      end
      % ----- Reduced force matrix ---------------------------------------
      fr=sparse([number_of_eigenvectors' Egv(:,number_of_eigenvectors)'*f]);
      % ----- time integration parameters -------------------------------
      ip=[dt total_time 0.25 0.5 10 max(number_of_eigenvectors) ntimes nhistr];
      % ----- time integration ------------------------------------------
      [Dsnapr,Dr,Vr,Ar]=step2b(kr,cr,mr,dr0,vr0,ip,fr,[]);
      % ----- mapping back to original coordinate system ----------------
      AR=Egv(nhist,number_of_eigenvectors)*Ar;
      DR=Egv(nhist,number_of_eigenvectors)*Dr;
      % ----- Saving data -----------------------------------------------
      if speed_increase == 0
              Max_acc = max(max(abs(AR')));
              Max_acc_vector = max(abs(AR'))';
              Max_disp = max(max(abs(DR')));
              Max_disp_vector = max(abs(DR'))';
              speed_start = speed;
      else
              Max_acc = [Max_acc max(max(abs(AR')))];
              Max_acc_vector = [Max_acc_vector max(abs(AR'))'];
              Max_disp = [Max_disp max(max(abs(DR')))];
              Max_disp_vector = [Max_disp_vector max(abs(DR'))'];
      end
    end
end
%-------------- Presenting results -------------------------------------
figure(5)
hastighet = linspace(speed_start*3.6,speed*3.6,length(Max_acc));
plot(hastighet,Max_acc,'b-')
xlabel('Velocity km/h')
ylabel('Maximum acceleration (m/s^2)')
hold on

figure(6)
hastighet = linspace(0,L_bridge,length(DR(:,1)));
plot(hastighet,-max(Max_acc_vector'),'m-')
xlabel('length (m)')
ylabel('Maximum acceleration (m/s^2)')
hold on
```

```
figure(7)
hastighet = linspace(speed_start*3.6,speed*3.6,length(Max_disp));
plot(hastighet,Max_disp,'-')
xlabel('Velocity km/h')
ylabel('Maximum displacement (m)')
hold on

figure(8)
hastighet = linspace(0,L_bridge,length(DR(:,1)));
plot(hastighet,-max(Max_disp_vector'),'k-.')
xlabel('length (m)')
ylabel('Maximum displacement (m)')
hold on


disp('Done')
```

B. MATLAB code - The distributed load model

```
%------------------------------------------------------------------------
% PURPOSE:
% Compute the dynamic response in a high-speed railway bridge.
% The load of the train are modelled as distributed loads.
%------------------------------------------------------------------------
% INDATA:
% number_of_element = The number of elements used to model the bridge
% L_bridge = Length of the bridge
% time_step_element = Number of time-steps per element
% damping = The damping value of the bridge
% omega1 = A constant used to when calculating the Rayleigh damping
% based on the natural frequency of the structure.
% omega2 = A constant used to when calculating the Rayleigh damping
% based on the natural frequency of the structure.
% lastutbredning = The extension of the distributed load.
% number_of_loads = The number of point loads used to model the distributed load
% E = Young's modoulous
% I = Moment of inertia
% m = The mass per meter
%------------------------------------------------------------------------
% AUTHOR: Andreas Gustafsson 2008-12-10
%
% Copyright (c) Andreas Gustafsson
%------------------------------------------------------------------------

clear all
close all
format long

for Load_model = 1:10
    for speed_increase=0:150
        %--------------------------- Indata ---------------------------

        number_of_element = 20;
        L_bridge = 10;
        time_step_element = 10;
        damping = 0.015;
```

```
omega1 = 75.234548162045442;
omega2 = 1.884955592153876e+002;
lastutbredning = 1.25;
number_of_loads=100;

% -------- Material data ------------------------------------------

E = 40.8e9;
I = 0.4632;
m = 34235*0.95;


%-------- Calculated parameters I -------------------------------------
A = 8.7875;
a0 = damping*2*omega1*omega2/(omega1+omega2);
a1 = damping*2/(omega1+omega2);
ep = [E A I m [a0 a1]];
load_on_steps = 10;
speed = 27 + speed_increase*0.5;
dx = L_bridge/(time_step_element*number_of_element);
dt = L_bridge/(speed*time_step_element*number_of_element);
time_passing_bridge = L_bridge/speed;
L_element = L_bridge/number_of_element;
dgf = (number_of_element+1)*3;
% ---------------------- Topology matrix ---------------------------
disp('Calculating element geometry')
[Ex,Ey,Edof]=geometry2(number_of_element,L_bridge);
disp('Done')
% ----------------------- Train model ---------------------------
disp('Reading train model')
[HSLM_length,F_train]=hslm_utbredd_load(Load_model,dx,lastutbredning,
number_of_loads);
disp('Done')
%-------- Calculated parameters II ----------------------------------
HSLM_time = HSLM_length/speed;
total_time = time_passing_bridge + max(HSLM_time) + load_on_steps*dt;
%-------- Stiffness, mass, force and damping matrix bridge -----------
disp('Assemble stiffness-, mass and damping matrix')
K = zeros(dgf);
M = zeros(dgf);
C = zeros(dgf);
for i=1:number_of_element
    [Ke,Me,Ce] = beam2d(Ex(i,:),Ey(i,:),ep);
    K = assem(Edof(i,:),K,Ke);
    M = assem(Edof(i,:),M,Me);
    C = assem(Edof(i,:),C,Ce);
end
disp('Done')
```

```
% --- Force matrix ---
f = zeros(dgf,(total_time/dt + 1));
[t,g] = gfunc([0 0;load_on_steps*dt -1],dt);
for k=1:length(HSLM_time)
    time_hist_HSLM = round(HSLM_time(k)/dt);
    if time_hist_HSLM == 0
        time_hist_HSLM = 1;
    end
    f(2,time_hist_HSLM : load_on_steps+time_hist_HSLM-1) = g(1:length(g)-1)
    + f(2,time_hist_HSLM : load_on_steps+time_hist_HSLM-1);
    for j=1:number_of_element
        global_u2 = 3*j-1;
        global_u3 = 3*j;
        global_u5 = 3*j+2;
        global_u6 = 3*j+3;
        time_hist_element = (j-1)*time_step_element;
        for i = 0:round(L_element/dx)-1
            %------------- Calculating parameters --------------------
            a = dx*i;
            b = L_element-dx*i;
            %------------- Calculating F matrix ---------------------
            f( global_u2 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps)) = F_train*b^2/L_element^2*(1+2*a/L_element)
            + f( global_u2 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps));
            f( global_u3 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps)) = F_train*a*b^2/L_element^2 + f( global_u3 ,
            (time_hist_HSLM + time_hist_element + i + load_on_steps));
            f( global_u5 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps)) = F_train*a^2/L_element^2*(1+2*b/L_element)
            + f( global_u5 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps));
            f( global_u6 ,(time_hist_HSLM + time_hist_element + i +
            load_on_steps)) = -F_train*a^2*b/L_element^2 + f( global_u6 ,
            (time_hist_HSLM + time_hist_element + i + load_on_steps));
        end
    end
end
% --- boundary condition, initial condition --------------------------
bc = [1 0; 2 0; (dgf-1) 0];
d0 = zeros(dgf,1);
v0 = zeros(dgf,1);
% --- output parameters ----------------------------------------------
ntimes=[0.1:0.1:1];
nhist=[2];
for i=1:number_of_element
    nhist = [[nhist] i*3+2];
```

```
        end
        % --- time integration parameters ------------------------------------
        ip=[dt total_time 0.25 0.5  10 10 ntimes nhist];
        % --- time integration ------------------------------------------------
        F=f;
        k = sparse(K);
        m = sparse(M);
        c = sparse(C);
        f = sparse(f);
        [Dsnap,D,V,A]=step2(k,c,m,d0,v0,ip,f,bc);
        if speed_increase == 0
            Max_acc = max(max(abs(A)));
        else
            Max_acc = [Max_acc max(max(abs(A)))];
        end
    end
%-------------- Presenting results -------------------------------------------
figure(5)
hastighet = linspace(27*3.6,speed*3.6,length(Max_acc));
plot(hastighet,Max_acc,'b-')
xlabel('Velocity km/h')
ylabel('Maximum acceleration (m/s^2)')
hold on

disp('Done')
```

C. MATLAB code - The interaction load model

```
%----------------------------------------------------------------------------
% PURPOSE:
% Compute the dynamic response in a high-speed railway bridge.
% The load of the train are modelled as distributed loads.
%----------------------------------------------------------------------------
% INDATA:
% m_fjader = mass of the load model
% k_fjader = stiffness of the load model
% number_of_element = The number of elements used to model the bridge
% L_bridge = Length of the bridge
% speed = velocity of the load model
% interpolation = Number of time-steps per element
% damping = The damping value of the bridge
% omega1 = A constant used to when calculating the Rayleigh damping
% based on the natural frequency of the structure.
% omega2 = A constant used to when calculating the Rayleigh damping
% based on the natural frequency of the structure.
% load_on_steps = Number of load on steps
% E = Young's modulus
% I = Moment of inertia
% m = The mass per meter
%----------------------------------------------------------------------------
% AUTHOR: Andreas Gustafsson 2008-12-10
%
% Copyright (c) Andreas Gustafsson
%----------------------------------------------------------------------------

hold on
clear all
%close all
format long

%--------------------------- Indata ------------------------------------

m_fjader = -170e3/9.81;
k_fjader = 170e6;
number_of_element = 20;
```

```
L_bridge = 10;
speed = 126;
interpolation = 4000; %18000
damping = 0.015;
omega1 = 75.234548162045442;
omega2 = 1.884955592153876e+002;
load_on_steps = 2;

% -------- Material data --------------------------------------------------

E = 40.8e9/2;
I = 0.4632;
m = 34235*0.95;

%-------- Fixed parameters -----------------------------------------------
A = 8.7875;
%-------- Calculated parameters ------------------------------------------
a0 = damping*2*omega1*omega2/(omega1+omega2);
a1 = damping*2/(omega1+omega2);
ep = [E A I m [a0 a1]];
dx = L_bridge/(interpolation*number_of_element);
dt = L_bridge/(speed*interpolation*number_of_element);
T = L_bridge/speed;
L_element = L_bridge/number_of_element;
dgf = (number_of_element+1)*3;
%-------- Load on soft ---------
[t,g] = gfunc([0 0;load_on_steps*dt -m_fjader*9.81],dt);
%-------- Stiffness, mass and force matrix fjder ----------------------------
f_fjader = [-m_fjader*9.81;0];    %ndrat
M_fjader = [m_fjader 0; 0 m_fjader];
K_fjader = [k_fjader -k_fjader; -k_fjader k_fjader];
K_fjader = sparse(K_fjader);
M_fjader = sparse(M_fjader);
% --------------------- Topology matrix ----------------------------------
disp('Calculating element geometry')
[Ex,Ey,Edof]=geometry2(number_of_element,L_bridge);
disp('Done')
%-------- Stiffness, mass, force and damping matrix bridge -------------------
disp('Assemble stiffness-, mass and damping matrix')
K = zeros(dgf);
M = zeros(dgf);
C = zeros(dgf);
for i=1:number_of_element
    [Ke,Me,Ce] = beam2d(Ex(i,:),Ey(i,:),ep);
    K = assem(Edof(i,:),K,Ke);
    M = assem(Edof(i,:),M,Me);
    C = assem(Edof(i,:),C,Ce);
```

```
end
k = sparse(K);
m = sparse(M);
c = sparse(C);
f = zeros(dgf,2);
disp('Done')
% --- boundary condition, initial condition ----------------------------------
bc = [1 0; 2 0; (dgf-1) 0];
d0 = zeros(dgf,1);
v0 = zeros(dgf,1);
bc_fjader = [2 0];
saved_bridge_displacement = zeros((number_of_element*3+3),1);
saved_bridge_acceleration = zeros((number_of_element*3+3),1);
% --- output parameters -------------------------------------------------------
ntimes = [0.1:0.1:1];
nhist=[1];
for i=2:dgf
    nhist = [[nhist] i];
end
% --- time integration parameters for bridge and spring ----------------------
ip=[dt dt 0.25 0.5  10 10 ntimes nhist];
ip_fjader = [dt dt 0.25 0.5  10 10 ntimes 1 2];
%----------------------------- Position 1 ------------------------------------
% ------------- Static analysis ---------------
[d0_fjader,F_start] = solveq(K_fjader,f_fjader,bc_fjader);
%---------------------- Calculated INDATA to Position 2 ----------------------
%--- Bridge response ---
d0 = zeros(dgf,1);
v0 = zeros(dgf,1);
F = zeros(dgf,2);
F_of_x = F_start(2);
%--- Spring response ---
f_fjader = zeros(2,2);
f_fjader(1,:) = -m_fjader*9.81;
%d0 is calculated in Static analysis
v0_fjader = [0 ;0];
%----------------------------- Positions j and i ---------------------------
for j=1:number_of_element
    global_u2 = 3*j-1;
    global_u3 = 3*j;
    global_u5 = 3*j+2;
    global_u6 = 3*j+3;
    for i = 1:L_element/dx
            %------------- Calculating parameters ----------------------------
            a = dx*i;
            b = L_element-dx*i;
            %------------- Calculating F matrix ------------------------------
```

```
        if i == 1 && j == 1
            F = zeros(dgf,7);
            F(2,1:load_on_steps) = g(1:length(g)-1);
            F(2,1+load_on_steps) = F_of_x;
            F(2,2+load_on_steps) = F_of_x*b^2/L_element^2*(1+2*a/L_element);
            F(3,2+load_on_steps) = F_of_x*a*b^2/L_element^2;
            F(5,2+load_on_steps) = F_of_x*a^2/L_element^2*(1+2*b/L_element);
            F(6,2+load_on_steps) = -F_of_x*a^2*b/L_element^2;
             %-------------- Bridge response ----------------------------
            ip=[dt (load_on_steps+1)*dt 0.25 0.5  10 10 ntimes nhist];
            [Dsnap,D,V,A]=step2(k,c,m,d0,v0,ip,F,bc);
            ip=[dt dt 0.25 0.5  10 10 ntimes nhist];
            F = zeros(dgf,2);
            D = D(:,load_on_steps+1:load_on_steps+2);
            A = A(:,load_on_steps+1:load_on_steps+2);
        elseif i == 1
            F( global_u2 ,1) = F( global_u5-3 ,2);
            F( global_u2-3 ,1) = 0;
            F( global_u3-3 ,1) = 0;
            F( global_u5-3 ,1) = 0;
            F( global_u6-3 ,1) = 0;
            F( global_u2 ,2) = F_of_x*b^2/L_element^2*(1+2*a/L_element);
            F( global_u3 ,2) = F_of_x*a*b^2/L_element^2;
            F( global_u5 ,2) = F_of_x*a^2/L_element^2*(1+2*b/L_element);
            F( global_u6 ,2) = -F_of_x*a^2*b/L_element^2;
             %-------------- Bridge response ----------------------------
            [Dsnap,D,V,A]=step2(k,c,m,d0,v0,ip,F,bc);
        else
            F( global_u2 ,1) = F( global_u2 ,2);
            F( global_u3 ,1) = F( global_u3 ,2);
            F( global_u5 ,1) = F( global_u5 ,2);
            F( global_u6 ,1) = F( global_u6 ,2);
            F( global_u2 ,2) = F_of_x*b^2/L_element^2*(1+2*a/L_element);
            F( global_u3 ,2) = F_of_x*a*b^2/L_element^2;
            F( global_u5 ,2) = F_of_x*a^2/L_element^2*(1+2*b/L_element);
            F( global_u6 ,2) = -F_of_x*a^2*b/L_element^2;
             %-------------- Bridge response ----------------------------
            [Dsnap,D,V,A]=step2(k,c,m,d0,v0,ip,F,bc);
         end
        saved_bridge_acceleration = [ saved_bridge_acceleration abs(A(:,2)) ];
        saved_bridge_displacement = [ saved_bridge_displacement abs(D(:,2))];
         %-- Calculating displacement in point x ---
          u2 = D( global_u2 ,2);
          u3 = D( global_u3 ,2);
          u5 = D( global_u5 ,2);
          u6 = D( global_u6 ,2);
          N2 = 1-3*a^2/L_element^2+2*a^3/L_element^3;
```

```
                 N3 = a*(1-2*a/L_element+a^2/L_element^2);
                 N5 = a^2/L_element^2*(3-2*a/L_element);
                 N6 = a^2/L_element*(a/L_element-1);
                 Displacement = N2*u2 + N3*u3 + N5*u5 + N6*u6;
            %------------- Spring response-------------------------------------
            bc_fjader = [2 Displacement];
            [ Dsnap_fjader , D_fjader , V_fjader , A_fjader ]=step2(  K_fjader ,
            [] , M_fjader , d0_fjader , v0_fjader , ip_fjader , f_fjader , bc_fjader );
            %-------- Calculating initial condition for next step --------------
            F_of_x = k_fjader*(D_fjader(1)-Displacement);
            d0 = D(:,2);
            v0 = V(:,2);
            d0_fjader = D_fjader(:,2);
            v0_fjader = V_fjader(:,2);
            %------------- Saving data ----------------------------------------
            saved_bridge_displacement = [ saved_bridge_displacement Displacement];
    end
end
%---------------- Arranging data -----------------------------------------------
temp = max(saved_bridge_acceleration')
temp2 = max(saved_bridge_displacement')
Acc = 0;
Disp = 0;
for i = 1:number_of_element
    Acc = [Acc temp(i*3+2)];
    Disp = [Disp temp2(i*3+2)];
end
%-------------- Presenting results ---------------------------------------------
figure(1)
x=linspace(0,10,length(Disp));
plot(x,-Disp)
xlabel('length (m)'), ylabel('displacement (m)')
title('Time step influence on displacement')
```

D. ADINA MODEL