Creation of a State-Space Model from a Finite Element Model for the active control algorithm efficiency tests.

N. Geffroy, L. Brunetti, B. Bolzon, A. Jeremie,
Laboratoire d'Annecy-le-Vieux de Physique des Particules
LAPP-IN2P3-CNRS
BP110, F-74941 Annecy-le-Vieux Cedex, FRANCE
Phone: +33450091600, Fax: +33450279495, contact person : E-mail: nicolas.geffroy@lapp.in2p3.fr

J. Lottin
Laboratoire Systèmes et Matériaux pour la MEcatronique
5 chemin de Bellevue
BP80439, F-74944 Annecy-le-Vieux Cedex, FRANCE
Phone: +33450096560, Fax: +33450276543

June 20, 2007

Abstract

This paper describes one part of the work done by the LAViSta group (Laboratories in Annecy working on Vibration and Stabilisation), dedicated to the modelling and simulation aspects.
The first chapter deals with the finite element modelling of the mechanical structure. This step is very important in order to obtain the most representative model. Thus the structure is entirely described by the mass, damping and stiffness matrices.
Then these matrices will be used to create the state-space model. The second chapter presents both the general method and the extended method required for LAViSta problems.
Finally practical examples are presented as well as the obtained results.

I. Introduction

The objective is to create a state-space model of the mechanical structure, from a finite element model given by the commercial software SAMCEF [1].

Therefore, using Matlab/Simulink [2], it will be possible to obtain the structure's dynamic response under different loads, which could be either forces due to a perturbation or an actuator, and/or a prescribed acceleration for instance.

Finally, thanks to this representative state-space model, simulation tests of active control efficiency and robustness can be realized.

II. Finite Element Modelling

1) The Standard Model

Finite element modelling is of prime importance, insofar as the finite element model (FE model) is required for the future results to be representative. In fact, the state-space model (St-Sp model) will be built according to the FE model, namely the mass (M), damping (B) and stiffness (K) matrices exclusively. Thus, the reader will have first to turn his attention to this point.

For a given structure, and with the help of commercial software such as SAMCEF, the FE model is quite simple to obtain. The general procedure (Design, material and behaviour assigning, mesh…) not being the subject of this paper, it will not be explicitly described in this paper. Nevertheless it finally consists in computing the matrices M, B and K, respectively the mass, damping and stiffness matrices. In this way the fundamental equation describing the dynamic behaviour of a structure discretized by FE is written:

$$M\ddot{q}(t) + B\dot{q}(t) + Kq(t) = g(t)\,, \qquad (1)$$

where the **q**(t) state vector collects the displacements of the structure by degree of freedom, while the **g**(t) vector indicates the corresponding applied loads.

2) Description of Damping

It is possible to compute the damping matrix B by using the Lord Rayleigh's hypothesis. It is probably one of the simplest ways to introduce damping.

This method consists in defining a viscous proportional damping of the form:

$$B = \alpha \cdot K + \delta \cdot M \tag{2}$$

with **α** and **δ** solutions of the following equations:

$$\varepsilon_i = \frac{1}{2}\left(\alpha \cdot \omega_i + \frac{\delta}{\omega_i} \right) \tag{3}$$

Thus two couples of values, namely the damping ratios ε_i and their corresponding circular frequencies ω_i ($\omega_i = 2\pi \cdot f_i$), are required for the damping matrix B to be defined. In a general way this method gives good results, although damping is over or under-estimated as a function of the frequency (see figure 1).

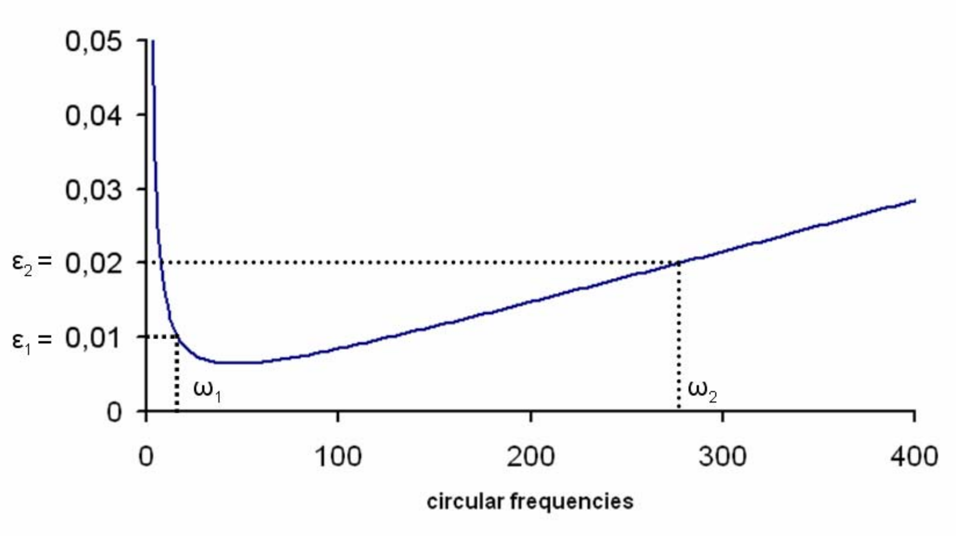

Figure 1: Damping ratio for Rayleigh's hypothesis as a function of ω

3) *The Use of Super Element*

The standard model presented above contains all the degrees of freedom (*dof*) of the discretized structure. Hence if the total number of *dof* is equal to **n**, the dimension of the state vector **q**(t) is equal to **n**, and the one of all matrices is **n*****n**. Moreover a FE analysis contains most of the time more than tens of thousand *dof*.

As a consequence the dynamic response computation can be extremely long. A way to overcome this point is to use the Super Element method. Indeed, if the desired results in terms of motion (acceleration, velocity, displacement) only concern specific location of the structure, it is possible to reduce the size of the system to solve.

Two different condensation methods are available in SAMCEF for a dynamic analysis, namely Guyan's method and Component-mode method (Craig & Bampton). The second one seems to be more reliable and really easier to use.

It consists in computing the constrained modes by keeping only *dof* which are desired in the results, that is to say in condensing all the others. The constrained modes are determined by assigning, successively, a unit displacement to each retained *dof*, all other retained *dof* being fixed.

Moreover, the influence of the condensed *dof* is nevertheless taken into account by computing the internal modes, corresponding to the eigen-modes of the same structure with the retained *dof* clamped. Then, these internal modes are introduced in the Super Element to improve its quality: the behaviour of the structure in the global system can be represented by superimposing the constrained modes and the internal modes.

Finally, for **r** retained *dof*, and **m** internal modes, the dimension of the M, B and K matrices is:

$$(\mathbf{r} + \mathbf{m}) * (\mathbf{r} + \mathbf{m}) \quad \text{with} \quad \mathbf{p = (r + m) << n}$$

Note that the number of **m** internal modes only depends on the difference between the eigenfrequencies of the structure (without Super Element) and the value of the first internal eigen-frequency (with Super Element). For further information the reader is referred to the book of J.F.Imbert [3].

III. The State-Space Model Creation

1) The general method

The state-space model is written according to equation (1), or its equivalent form in the case of a Super Element (square matrices of **p** dimension). The state-space model will have the following form, assuming that only external forces can be applied to the model:

$$\begin{cases} \dot{x} = Ax + Bu \\ y = Cx + Du \end{cases}$$

Where the state vector **x**, the input vector **u** and the output vector **y** are defined as:

$$x = \begin{Bmatrix} q \\ \dot{q} \end{Bmatrix} \qquad u = \{F\} \qquad y = \begin{Bmatrix} q_{out} \\ \dot{q}_{out} \\ \ddot{q}_{out} \\ F_{out} \end{Bmatrix} \qquad (4)$$

Furthermore, assuming that the number of states is equal to **2p**, the number of inputs is **in** and the one of output is **out**, the matrices A, B, C and D have the following characteristics, as illustrated on figure 2.

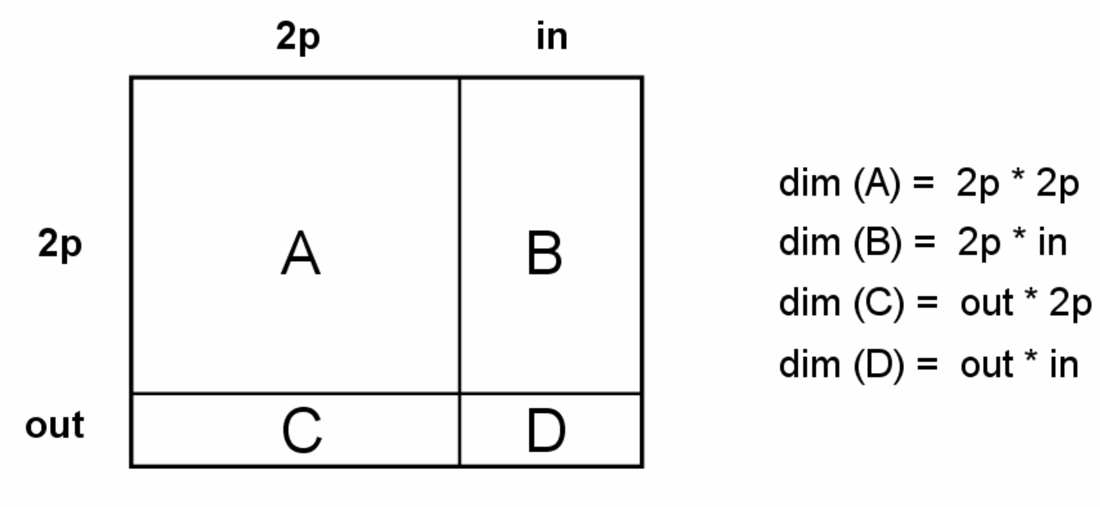

Figure 2: Characteristic dimensions of the state-space matrices.

According to equation (1), the acceleration vector can be written as follows:

$$\ddot{q} = -M^{-1}B\dot{q} - M^{-1}Kq + M^{-1}F \qquad (5)$$

Finally, the different matrices are defined below, assuming that only external forces can be applied to the model:

$$A = \begin{bmatrix} 0 & I \\ -M^{-1}K & -M^{-1}B \end{bmatrix} \qquad B = \begin{bmatrix} 0 \\ M^{-1} \end{bmatrix}$$

$$C = \begin{bmatrix} I & 0 \\ 0 & I \\ -M^{-1}K & -M^{-1}B \\ 0 & 0 \end{bmatrix} \qquad D = \begin{bmatrix} 0 \\ 0 \\ M^{-1} \\ I \end{bmatrix} \qquad (6)$$

As a conclusion, a Matlab program for instance would be able to read the matrices of the FE model, and convert them into a state-space model following the method described above. Then, this kind of model integrated in Simulink will give the dynamic response of the modelled structure under one or several inputs, which are only pinpoint forces.

2) The Extended method for the LAViSta problems

The external disturbances acting on the structure are not always point forces. For example, a structure can be disrupted by the ground motion. This is exactly the case of the LAViSta experimental set up, which is a slender free-fixed structure. Indeed, vibrations of the mock-up are due both to the acoustic pressure along the structure (see presentation done during ILC 2006 conference [4]), and to a prescribed acceleration. This acceleration comes from the ground motion and acts on the structure via its clamping.

In the general method inputs are assumed to be only pinpoints forces. That is the reason why it is not suitable to take into account the prescribed motion. To overcome this problem, the following solutions can be adopted.

In a FE code, the dynamic response of a structure under prescribed acceleration $\ddot{q}_2(t)$ is computed according to the following way. First, the corresponding prescribed velocities and displacements are numerically integrated:

$$\dot{q}_2(t) = \int_0^t \ddot{q}_2(\tau)\cdot d\tau \qquad \text{and} \qquad q_2(t) = \int_0^t \dot{q}_2(\tau)\cdot d\tau \tag{7}$$

Then the system to solve is now written:

$$\begin{bmatrix} M_{11} & M_{12} \\ M_{21} & M_{22} \end{bmatrix} \begin{Bmatrix} \ddot{q}_1 \\ \ddot{q}_2 \end{Bmatrix} + \begin{bmatrix} B_{11} & B_{12} \\ B_{21} & B_{22} \end{bmatrix} \begin{Bmatrix} \dot{q}_1 \\ \dot{q}_2 \end{Bmatrix} + \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix} = \begin{Bmatrix} g_1(t) \\ -r_2(t) \end{Bmatrix} \tag{8}$$

where the index 2 stands for the *dof* where acceleration is applied and index 1 for all other *dof* of the model. Moreover, $g_1(t)$ represents the possible external point forces.

The first equation of the system (8) allows the dynamic response computation, provided one carries over the right hand side forces of inertia, dissipation and stiffness associated with the prescribed motion:

$$M_{11}\cdot\ddot{q}_1 + B_{11}\cdot\dot{q}_1 + K_{11}\cdot q_1 = g_1(t) - M_{12}\cdot\ddot{q}_2 - B_{12}\cdot\dot{q}_2 - K_{12}\cdot q_2 \tag{9}$$

Hence, in a general way, the input vector **u** of the state-space model will put together the terms of point forces and the terms of forces of inertia, dissipation and stiffness, which corresponds to the right hand side terms of equation (9).

Consequently, if a prescribed acceleration is applied, dimensions of A, B, C and D are fixed as illustrated by figure 3, and this whatever the number of point forces as additional input. The B and D matrices become bigger in this case. The only parameter is from now on the number of outputs **out**.

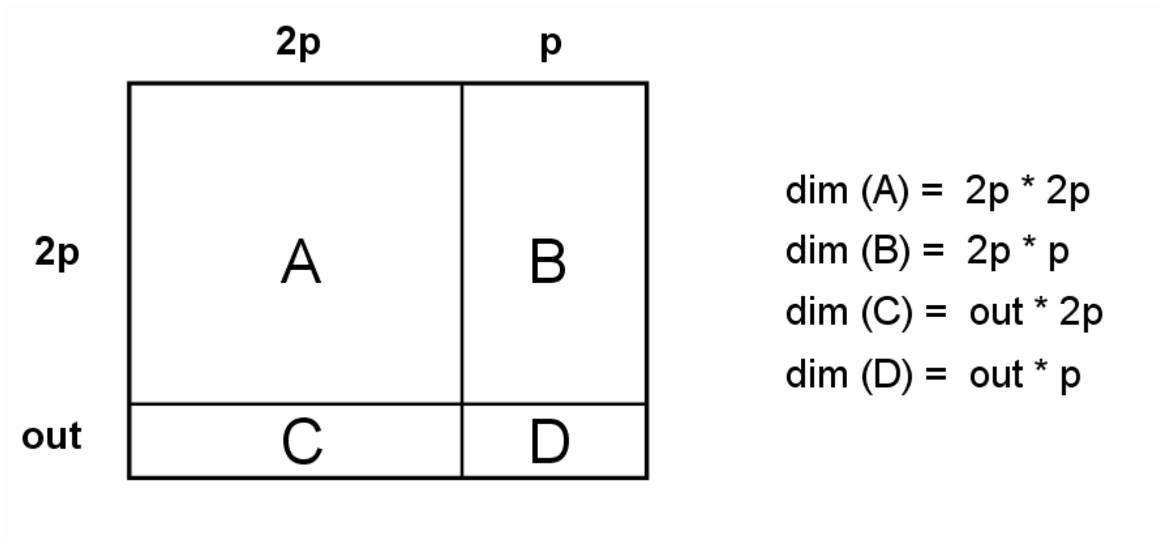

Figure 3: Characteristic dimensions of the state-space matrices in case of prescribed acceleration

Practically, to have access to the additional forces (inertia, dissipation and stiffness), that is to say the three terms M_{12}, B_{12} and K_{12} , it is possible not to fix in the FE model the *dof* which are clamped in reality. Thus the previous terms will explicitly appear in the matrices M, B and K. However, once used for the definition of the input vector **u**, these terms have to be deleted from M, B and K, to get M_{11}, B_{11} and K_{11}.

IV. Practical examples of state space model use in Simulink

The generic form of state-space model representation with Simulink is shown below (figure 4).

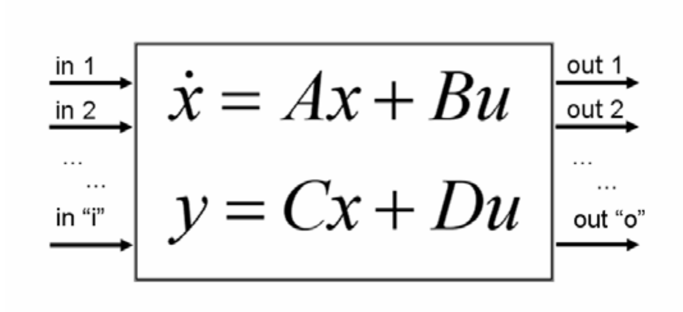

Figure 4: Block representation of the state-space model in Simulink

This simple representation allows a simple use of the model and an easy calculation of the response. Note that in the following examples, the state-space model is discretized.

1) Simple dynamic response computation

In order to illustrate the possibilities given by such a model, figure 5 shows the block representation realized with Simulink.

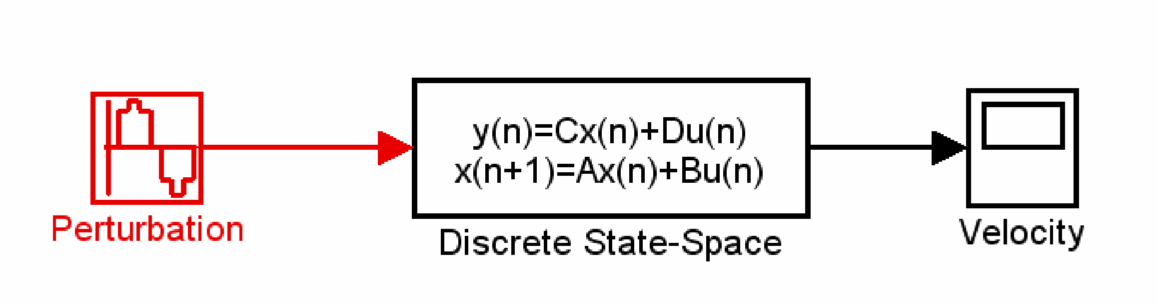

Figure 5: Simulink representation for calculation of the process response

The state-space model represents a free-fixed structure, for which a sinusoidal load (called “Perturbation” on figure 5) is applied at the free extremity of the structure (single input). As a result the velocity of this point (single output) is presented on figure 6.

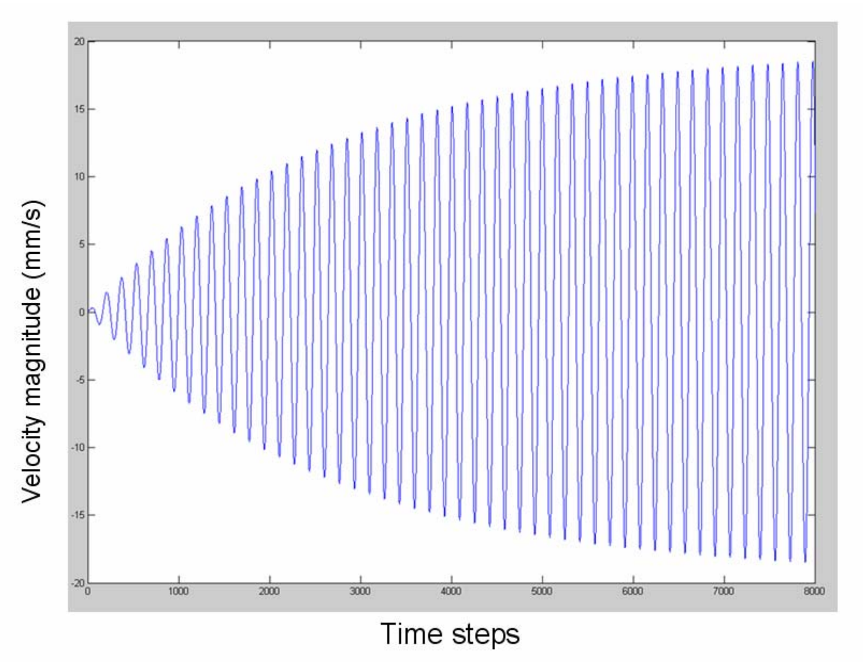

Figure 6: Velocity of the free extremity of the structure as a function of time

Figure 6 describes the response of the structure under sinusoidal force, whose frequency corresponds to its first eigenfrequency. As a result a resonance phenomenon can be observed. The response has a sinusoidal shape (of same frequency), whose magnitude is given by the damping ratio determined by Figure 1. For a longer simulation, a steady state would be observed.

2) Dynamic response associated with active control

The objective of the study was to be able to predict the dynamic response of a given structure, under perturbations like loads or prescribed acceleration, and especially to check the efficiency and the robustness of the active control algorithm. The computation process is presented in Figure 7. The rejection method and the algorithm will not be explained in this paper, but can be found in the LAViSta paper [5].

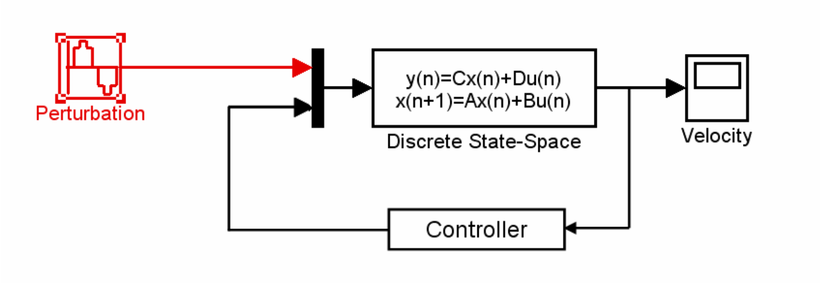

Figure 7: Simulink representation of the process' response calculus, including the controller

As an illustration, figure 8 shows the beginning of the active control of the free-fixed structure, as a function of time. Note that the motion of the structure is still due to a sinusoidal load (resonance frequency) and that, for reasons of understanding, the controller is activated when the process is in steady state. Indeed, without active control, the motion would be the same as long as there is a perturbation force. As a result of the active rejection, the velocity magnitude decreases.

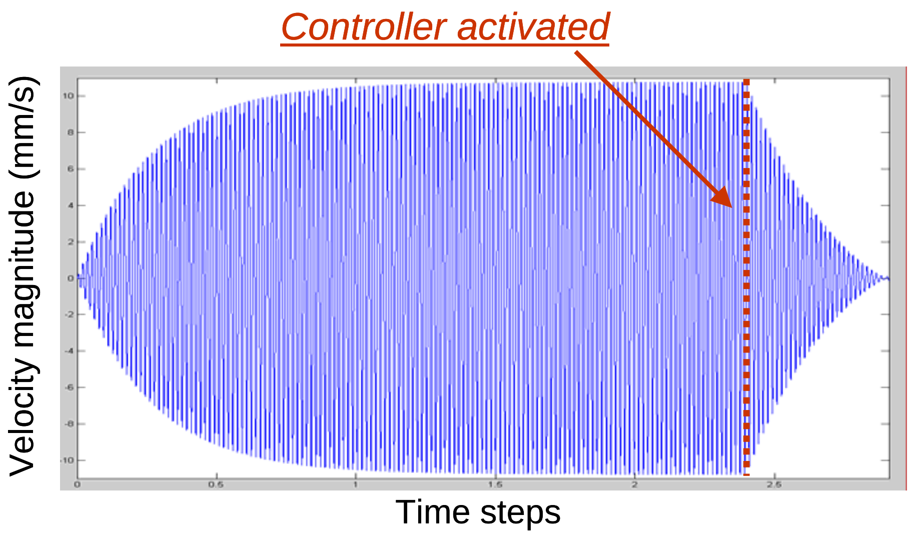

Figure 8: Velocity of the free part, without and with active control, as a function of time.

The above result illustrates the simulation of the whole system, namely dynamic response and active control. The aim of the present paper was to explain how to realize a state-space model especially in order to test accurately the active control algorithm. For further information concerning the latter and to see vibration rejection results, the reader is referred to the presentation done during ILC 2007 conference at DESY [6].

V. Conclusions and future prospects

Note that the computation of dynamic response of structures can be achieved directly in the FE code. Typically the system corresponding to the dynamic equations can be solved either by direct integration (using integration schemes as Newmark, HHT…) or by modal superimposition. It is also possible to link the calculation of the response with the active control algorithm (from Matlab) by translating the program with the help of a FORTRAN compiler.

Nevertheless, computing the response with Simulink (through a state space model) is much more advantageous. Indeed, most of the time, once the FE model is done, it is left unmodified, whereas it is often necessary to fit some parameters of the algorithm and to check their influence on the response. Doing this with Simulink is much simpler and faster, particularly since the evolution of almost every rejection parameters can be displayed. On the contrary, all of them cannot be monitored with Samcef. Note however that if stresses and strains are required, Samcef is still the most efficient.

Concerning the future prospects, given the motion is available everywhere along the structure (as many outputs as wanted), and given applying a load is as well possible everywhere on the structure (as many inputs as wanted), a new study can be considered: multi-sensors and multi-actuators techniques. Moreover it would also be interesting to add a model of the(s) actuator(s), through its transfer function for instance, in order to have an even more representative model, and to get more accurate results.

References

[1] http://www.samcef.com

[2] http://www.mathworks.fr/

[3] "Analyse des structures par éléments finis", Cépaduès Edition, J.F. Imbert

[4] "Direct impact of acoustic noise on the vibrations of a free-fixed beam", ILC 2006 workshop, 15/17 may 2006, LAL, Orsay, http://ilcagenda.linearcollider.org/getFile.py/access?contribId=39&sessionId=16&resId=1&materialId=slides&confId=293

[5] "Status report on active stabilisation of a linear collider final focus quadrupole mock up", L. Brunetti and al., EUROTeV-Report-2006-097.

[6] "Active stabilisation of a mechanical structure", ILC 2007 Workshop, 2007/06/01, DESY, http://ilcagenda.linearcollider.org/contributionDisplay.py?contribId=288&sessionId=90&confId=1296

Acknowledgements

This work is supported by the Commission of the European Communities under the 6th Framework Programme "Structuring the European Research Area", contact number RIDS-011899.