

# Model Reduction application to Harmonic pre-stressed of a Electromechanical model

Laura Del Tin, 2006, email: [ldeltin@deis.unibo.it](mailto:ldeltin@deis.unibo.it)

## **Introduction**

This tutorial shows how to apply Model Order Reduction to the harmonic pre-stressed analysis of electromechanical structures. The necessary procedure is illustrated step by step. More information about the theoretical basis and the computational advantages are reported in:

L. Del Tin, R. Gaddi, A. Gnudi, E. B. Rudnyi, A. Greiner, J. G. Korvink, "Efficient pre-stressed harmonic analysis of RF-microresonators by means of model order reduction.", EuroSimE 2006, Thermal, Mechanical and Multiphysics Simulation and Experiments in Micro-Electronics and Micro-Systems, Como (Milano), Italy, 24-26 April 2006.

Preprint at:

<http://www.imtek.de/simulation/>

## **Files:**

`model.ans` is an ANSYS script to build the model

`matrices.ans` is an ANSYS script to write files needed for system matrices generation

`harmonic.ans` is an ANSYS script to run harmonic pre-stressed analysis.

`plots.nb` is a Mathematica notebook which allows to build system matrices for the full model, to simulate the reduced model and to compare its results with ANSYS full model results.

## **Background for electromechanical model:**

The electromechanical model of a mechanical microresonator is used in this tutorial. The device is a free-free beam vertical microresonator with electrostatic actuation. Geometry, material properties and dimensions has been taken from the article:

K.Wang,A-C.Wong,C.T-C.Nguyen,"VHF Free-free beam high-Q micromechanical resonators", IEEE J.of Microelectromechanical system, vol.9, n.3, September 2000

Preprint at:

<http://www.eecs.umich.edu/~ctnguyen/publications.ctnguyen.htm>

Both electrical and mechanical quantities are of interest for this kind of device. For the computation of current, not included in the set of d.o.f., some extra steps are required for output matrix creation and results post-processing.

## **Step1.Building the model**

Run the script `model.ans` in batch mode ore from ANSYS GUI.The script will produce `geo.db` ANSYS database for the model.

## **Step 2.Running Harmonic Pre-Stressed Simulation in ANSYS**

This step is only required to compare ANSYS results with the one obtain with the reduced system.

Run the script `harmonic.ans`. It requires the `geo.db` file from step 1 in the current directory. The script perform a static non-linear analysis, followed by a harmonic analysis for different frequency values (a number "nstep" of frequency spanning logartimically the range "freqmin"- "freqmax").For each frequency, the vertical displacement for the chosen output node and the total current flowing in the device are written in the `results.txt` file in Matrix Market format, while the file

results.txt.names contains output names.

### **Step 3. Creating files for system matrix generation**

System matrices must be derived from the assembled tangent stiffness matrix.

For the system considered, the real part of this:  $\Re\{\bar{K}\} = K - \omega^2 M$

Computing this matrix for  $\omega=0$  we can derive directly the stiffness matrix K. Another frequency is then needed for the computation of M. The value of  $10^{13}$  is chosen to gap the difference in orders of magnitude between the elements of K and M matrices, which leads to numerical error in M calculation.

If air-damping is not considered, the imaginary part is null. The damping matrix E contains in this case only the equations for current computation and has not null elements only in rows corresponding to constrained voltage d.o.f. which are eliminated by dirichlet boundary conditions application. The information for current calculation need to be extracted from the original matrix, and are used to compute the contribution to current of voltage derivative before system reduction and the output matrix for the contribution to current of displacement derivatives. The two values needs then to be summed and multiplied by  $i\omega$  to obtain the current. The current output can be also added to other mechanical output in a single output matrix.

Run the script matrices.ans to create files needed for system matrices generation. The following files are created:

- filefreq0.full, filefreq0.emat, filefreq0.esav from which is possible to extract the complex tangent stiffness matrix that ANSYS uses to perform harmonic simulation for  $\omega=2*\pi*f=0$
- filefreq1e13.full, filefreq1e13.emat, filefreq1e13.esav from which is possible to extract the complex tangent stiffness matrix that ANSYS uses to perform harmonic simulation for  $\omega=2*\pi*f=10^{13}$

Run the following commands to extract complex tangent stiffness matrix at the two frequencies:

```
dumpmatrices filefreq0.full -C outdof.txt -w mat0
```

```
dumpmatrices filefreq1e13.full -C outdof.txt -w mat1e13
```

The output files are matrices written in Matrix Market format. mat0 and mat1e13 are the base names of the output files, which have extention .K for the real part of tangent stiffness matrix, and .K.im for immaginary part. Extensions .B and .C are used for load vector and output matrix (both real).

Run the following command to extract damping matrix E of the original system, and output matrix C needed for current computation:

```
dumpmatrices -f filefreq0.full filefreq0.emat -o -C output.txt -w emat
```

File emat.K, emat.E, emat.C, emat.M, emat.S, emat.B, emat.C.names are created, of which only emat.E and emat.C will be used.

### **Step 3. System matrix generation**

Matrices extracted in step 2 will be processed in mathematica for system matrix generation. This is done by the first part of notebook plots.nb.

- Making output matrix from emat file:

The matrices emat.E and emat.C are readed and used to compute the entry of the output matrix

for current due to displacement derivatives calculation and the voltage derivatives contribution to current. At this aim the number of voltage nodes with non zero dirichlet boundary condition has to be known.

- Making system matrices:

The matrices `mat0.K` and `mat1e13.K` are imported and used for K and M computation. Matrices `m0.B` (equal to `m1e13.B`) and `m0.C` are then imported and the current output is added to the latter. A dynamic system object is then created and printed on file `full.*`.

#### ***Step 4. Model order reduction***

Run the following command to reduce the system of matrices `full.*`:

```
mor4ansys -MM full -s UMFPACK -t 1e-20
```

A 30 d.o.f. reduced order model is created and written in Matrix Market format in files `mor.*`.

#### ***Step 5: Simulation of the reduced system and plotting:***

Harmonic analysis of the reduced system is made in mathematica. The second part of notebook `plots.nb` reads the system, simulate it, transform its current results, and then plots results together with the one read from ANSYS.