



**UNIMORE**  
UNIVERSITÀ DEGLI STUDI DI  
MODENA E REGGIO EMILIA



Dipartimento di Ingegneria  
“Enzo Ferrari”

# Progettazione Assistita di Organi di Macchine

Sara Mantovani  
[sara.mantovani@unimore.it](mailto:sara.mantovani@unimore.it)

# Agenda

Goal and nomenclature

Model setup

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- Considering damping effect

References

# Agenda

## Goal and nomenclature

Model setup

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

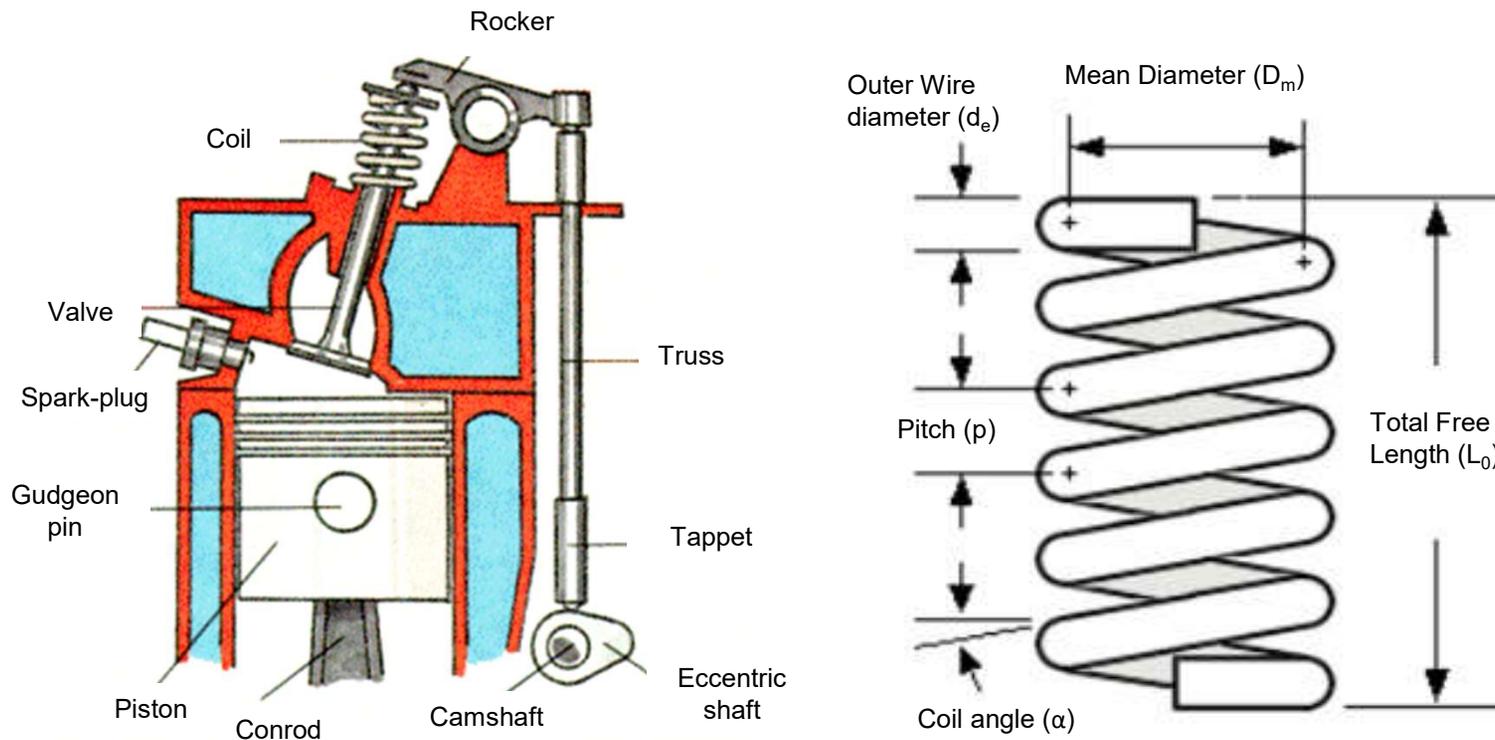
- Neglecting damping effect
- Considering damping effect

References

# Goal and nomenclature

## Distribution coil dynamic response

The present lesson aims to evaluate the natural frequencies and the frequency response occurring in a helical coil applied to a engine system distribution.



# Goal and nomenclature

## Distribution coil dynamic response

The present lesson aims to evaluate the modal and the frequency response occurring in a helical coil applied to a engine system distribution.

Mean diameter and mean radius:

$$D_m = 40 \text{ mm}, R_m = 20 \text{ mm}$$

Wire outer diameter:  $D_e = 12 \text{ mm}$

Wire inner diameter:  $D_i = 6 \text{ mm}$

Coil pitch:  $p = 15 \text{ mm}$

Total coils:  $n_t = 4,5$

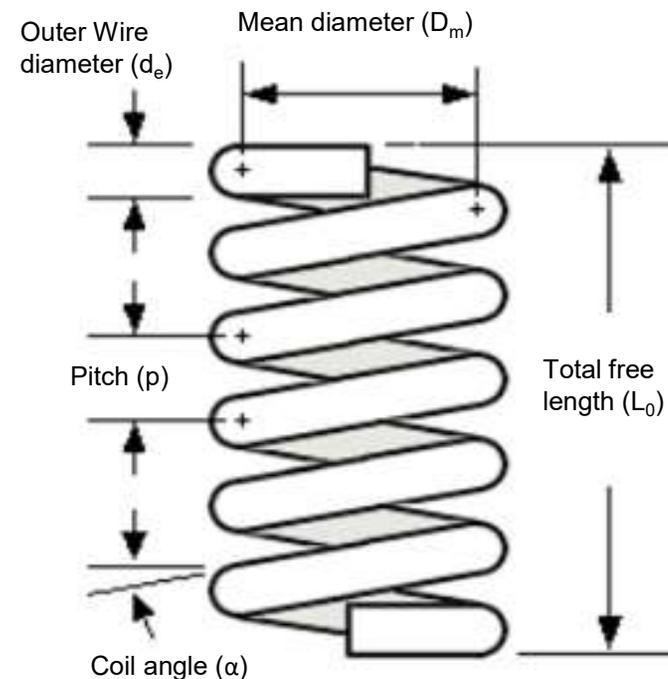
Total length:  $L_0 = 67.5 \text{ mm}$

Material properties

Titanium:

$$E = 110 \text{ GPa}, \nu = 0.3,$$

$$\rho = 4.7 \cdot 10^{-9} \text{ t/mm}^3$$



# Goal and nomenclature

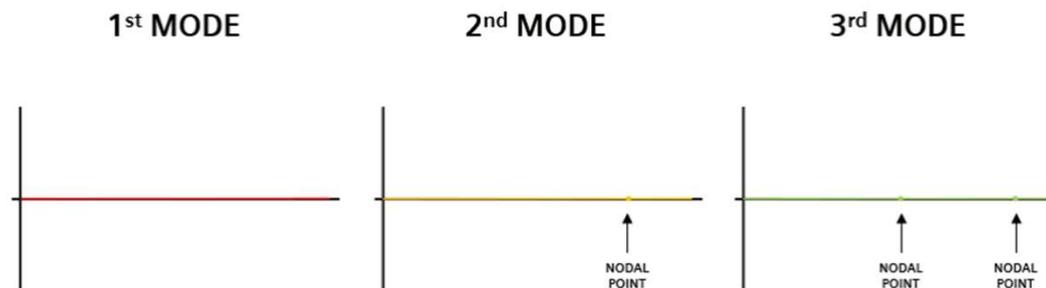
## Modal analysis

The **first mode** or natural frequency is the one at which the component will vibrate after all external excitations are removed. Additional natural frequencies represent the oscillation of the components in other deformed shaped or «modes».

Modal vibration only occurs when the part is being shaken at a frequency which is near a natural frequency.

Without **damping**, an oscillating body is kept in motion forever. Damping represents the inefficiencies of the material due to energy loss at a molecular level or of the system due to the component interaction. Higher damping factors cause the oscillation's amplitude to decrease so the component slowly ( or not so slowly) stabilized.

Knowing the natural frequencies of a design subject to harmonic inputs is important. When a part is excited at a frequency it is «comfortable» vibrating at, the effects of the input are magnified and any cause premature or catastrophic failure.



# Goal and nomenclature

## Modal analysis

The goal of a modal study is to ensure that the system does not have a resonant frequency near to the operating frequency or in the range of operating frequencies. If the first mode is lower than the operating speed, the product user will notice a «shudder» on start-up as the speed passes that frequency. To ensure that resonance effects are avoided within the operating frequency range, certain references suggest that natural frequencies occur only below one-third of the minimum operating frequency and above three times the maximum.

A modal analysis provides an understanding of the induced mode shapes themselves. Deformation patterns at an operating frequency may be deemed acceptable if they do not affect an inherently weak section of the system.

A modal analysis does not require BCs but may utilize a constraint case if constraints are present. If the model is not constrained fully in all six DOFs, the first modes will correspond to rigid body motion in each of the unconstrained directions with a frequency of approximately zero. These are called «rigid body modes» and should in theory be of zero magnitude, although numerical round-off may give them a small nonzero value.

# Goal and nomenclature

## Modal analysis

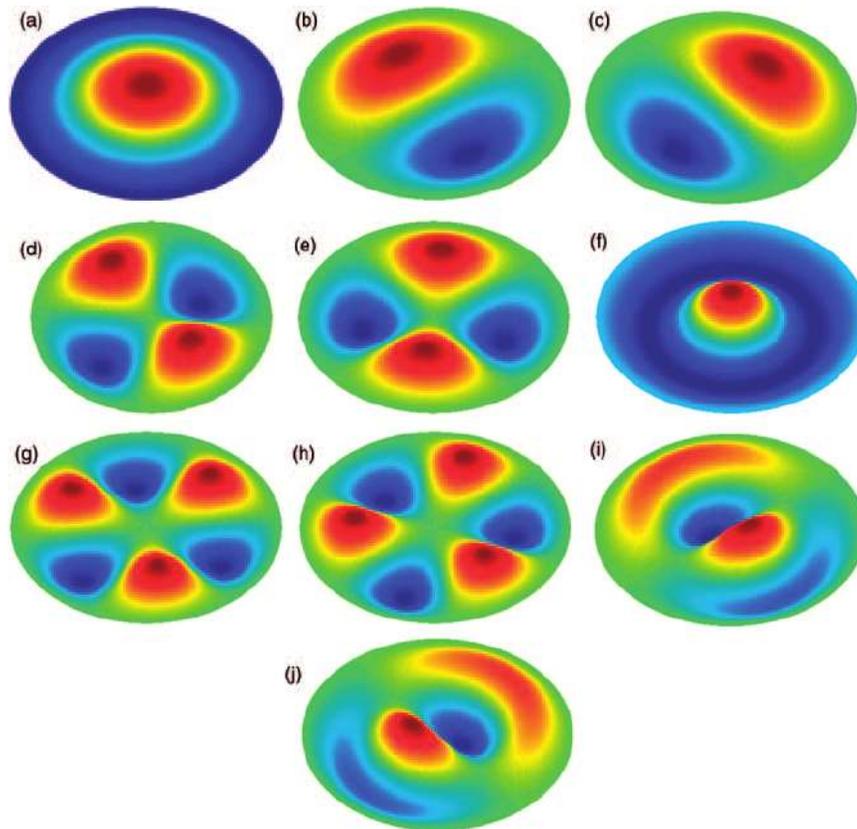
Some solvers (e.g. mentat) require you to specify that an unconstrained model is unconstrained before initiating the modal study. An overconstrained model will behave too stiffly and, therefore, results in an overprediction of the first modes.

In most cases, a first or second mode near an operating frequency will cause noticeable vibration amplitude. Adjusting your geometry to move the natural frequency is somewhat of an art. The natural frequency of a part is related to its weight and its stiffness. However, many techniques for increasing stiffness also add weight. The proper combination of increased stiffness, reduced weight, and redistributed weight is required to fine tune natural frequency.

The danger of using symmetry is that there are many more skew-symmetric mode shapes for a general structure than symmetric. If symmetry is used, the only mode shapes will correspond to the specific symmetry constraints. If the frequency of interest corresponds to a skew-symmetric mode shape in a full model, an important result will be missed in a symmetric model.

# Goal and nomenclature

## Modal analysis



The first 10 mode shapes for a fully clamped circular disk

Symmetric and skew-symmetric modes are present.

# Agenda

Goal and nomenclature

## Model setup

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- Considering damping effect

References

# Mesh Generation

## Coil shape definition

The coil will be built starting from crucial node and points proper of this component, by the MESH GENERATION menu.

The screenshot displays the MESH GENERATION software interface. On the left is a menu with various options:

- MESH GENERATION
  - NODES: ADD, REM, EDIT, SHOW
  - ELEMS: ADD, REM, EDIT, SHOW
  - PTS: ADD, REM, EDIT, SHOW
  - CRVS: ADD, REM, EDIT, SHOW
  - SRFS: ADD, REM, EDIT, SHOW
  - SOLIDS: ADD, REM, SHOW
  - BETWEEN NODE: BETWEEN POINT
  - ELEMENT CLASS: ▾ QUAD (4)
  - CURVE TYPE: ▾ LINE
  - SURFACE TYPE: ▾ QUAD
  - SOLID TYPE: ▾ BLOCK
  - COORDINATE SYSTEM
    - SET: ▾ RECTANGULAR, GRID
  - CLEAR MESH: CLEAR GEOM
  - ATTACH: AUTOMESH
  - CHANGE CLASS: CHECK
  - CONVERT: DUPLICATE
  - ELEMENT TYPES: EXPAND
  - INTERSECT: MOVE
  - RELAX: RENUMBER
  - REVOLVE: SOLIDS
  - STRETCH: SUBDIVIDE
  - SWEEP: SYMMETRY
  - ALL: SELEC, VISIB, OUTL, TOP
  - EXIST: UNSEL, INVIS, SURE, BOT
  - SELECT: SET, END LIST (#)
  - RETURN: MAIN

The main 3D view shows a coordinate system with the following nodes and points defined:

- O (0,0,0) - Origin point, marked with a pink plus sign.
- Q (0,0,67.5) - Max axial coil extent, marked with a pink plus sign.
- NODE ID1 (20,0,0) - A point marked with a pink square.

The bottom toolbar contains the following options: UNDO, SAVE, DRAW, FILL, RESET VIEW, TX+, TY+, TZ+, RX+, RY+, RZ+, ZOOM BOX, IN, SHORTCUTS, UTILS, FILES, PLOT, VIEW, DYN. MODEL, TX-, TY-, TZ-, RX-, RY-, RZ-, OUT, SETTINGS, HELP.

# Mesh Generation

## Coil shape definition

```

-----
| Define the points located at:
| - the system origin O (0,0,0)
| - axial extension of the coil along the Z-axis that it is equal to L = p * n
|-----
*add_points
0
0
0
0
0
0
4.5*15
|-----
| Define the position of the coil mean radius by a node, ID 1.
|-----
*add_nodes
20
0
0
0
|

```

CLEAR MESH	CLEAR GEOM			
ATTACH	AUTOMESH			
CHANGE CLASS	CHECK			
CONVERT	DUPLICATE			
ELEMENT TYPES	EXPAND			
INTERSECT	MOVE			
RELAX	RENUMBER			
REVOLVE	SOLIDS			
STRETCH	SUBDIVIDE			
SWEEP	SYMMETRY			
ALL	SELEC	VISIB	OUTL	TOP
EXIST	UNSEL	INVIS	SURE	BOT
SELECT	SET	END LIST (#)		
RETURN	MAIN			

O (0,0,0)

NODE ID1 (20,0,0)

Q (0,0,67.5)  
Max axial coil extent

ME Software

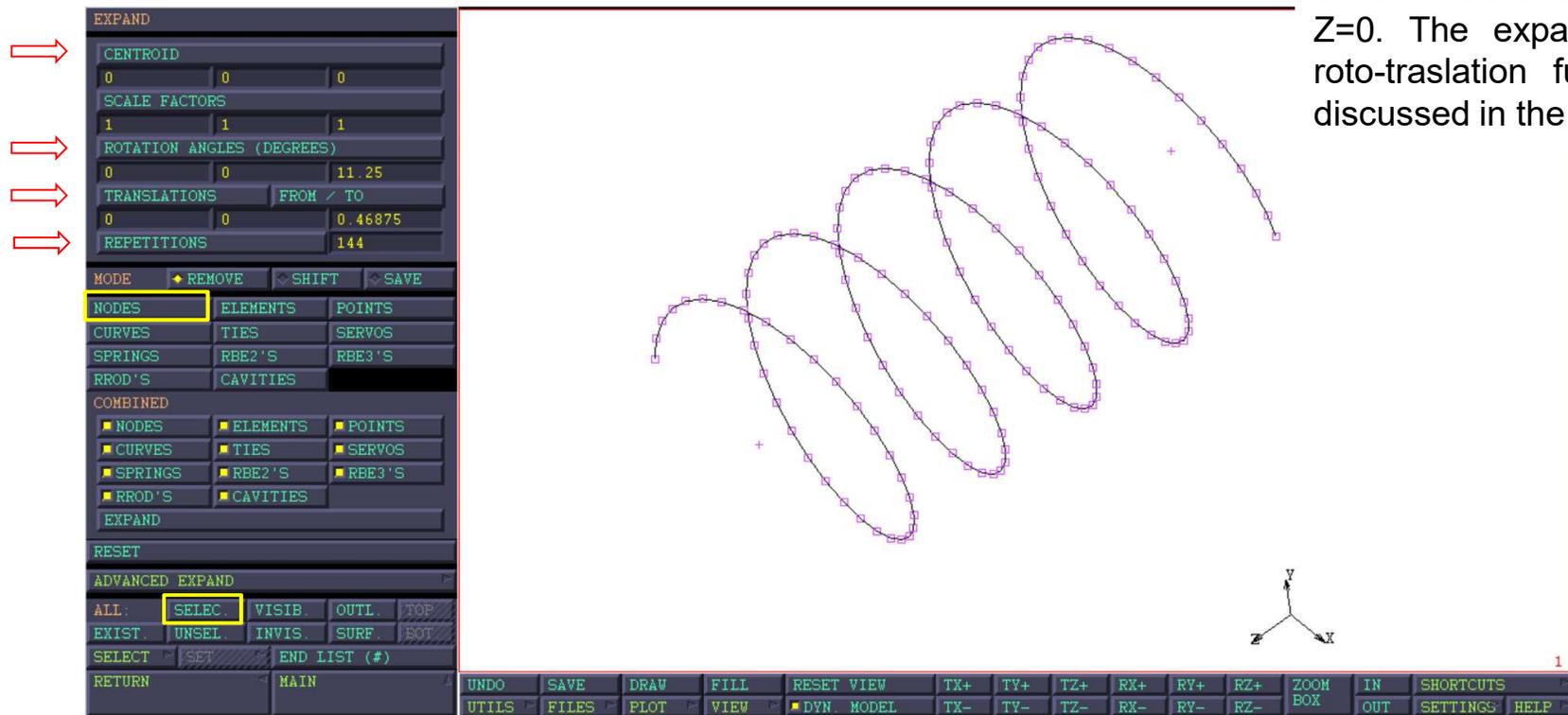
1



# Mesh Generation

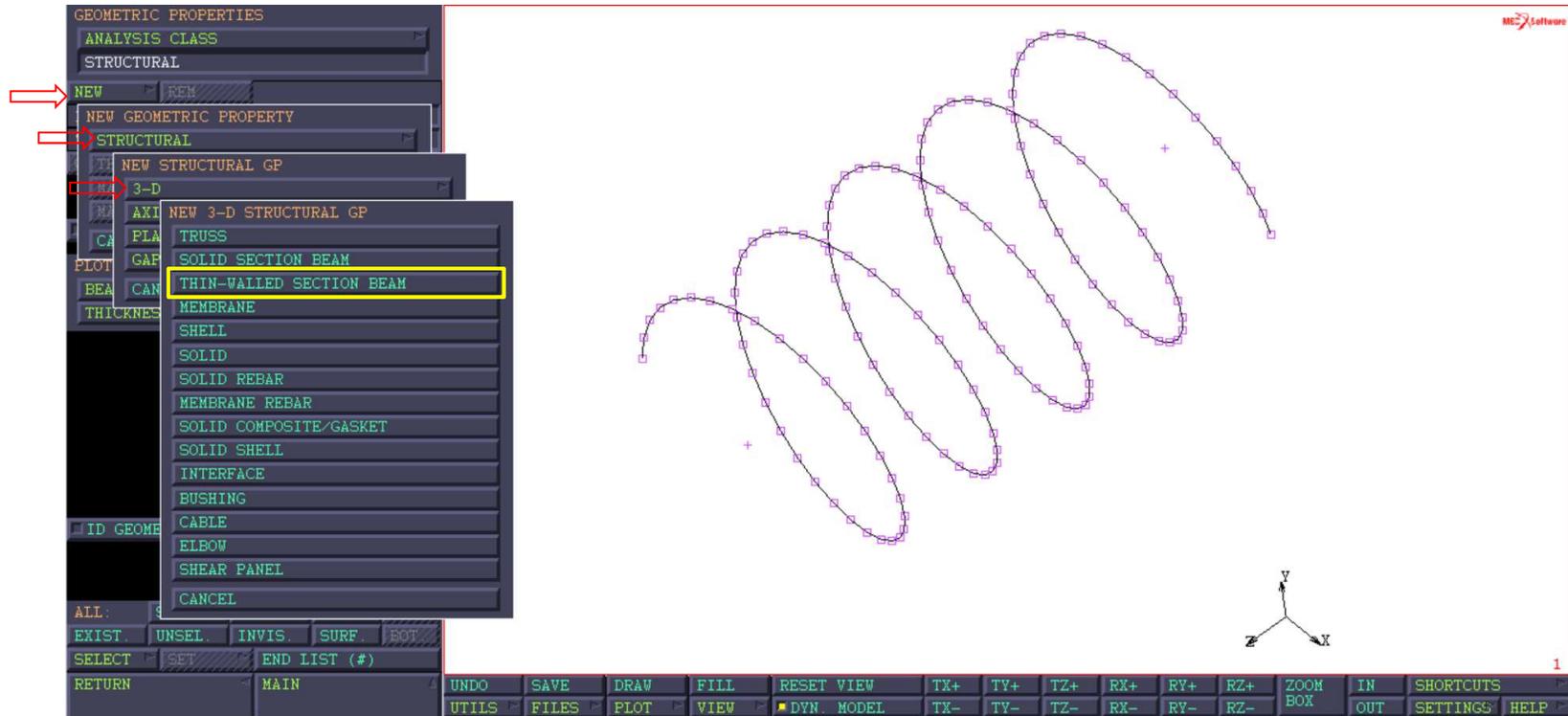
## Coil shape definition: expand

The coil is modelled by 1D line elements, obtained from the expansion of the NODE ID1, at the mean radius of the coil lying on the XY reference plane at  $Z=0$ . The expansion is a roto-traslation function as discussed in the following.



# Geometric properties

## Hollow circular cross-section



The coil geometric property is defined for thin-walled section beam, where the hollow circular cross section is assessed setting:

- The wire mean radius ( $r$ )  
$$r = (d_e + d_i) / 4 = (12 + 6) / 4 = 4.5 \text{ mm}$$
- The wire wall thickness is:  
$$t = (d_e - d_i) / 2 = (12 - 6) / 2 = 3.0 \text{ mm}$$

# Geometric properties

## Hollow circular cross-section

GEOMETRIC PROPERTIES

ANALYSIS CLASS

STRUCTURAL

NEW REM

NAME de\_12\_di\_6

TYPE mech\_three\_beam\_gen

COPY PREV NEXT EDIT

PROPERTIES

BEAM SECTIONS TABLES

PLOT SETTINGS

BEAM SHELL

THICKNESS DIRECTION

STRUCTURAL 3-D THIN-WALLED SECTION BEAM PROPERTIES

ELEMENT TYPES LINE (2) 78 CONNECTION

LINE (3) 76

CROSS SECTION CIRCULAR

RADIUS 4.5

WALL THICKNESS 3

ORIENTATION (LOCAL ELEMENT COORDINATE SYSTEM)

TYPE Local Z-axis along element

VECTOR DEFINING LOCAL ZX-PLANE

COMPONENTS IN GLOBAL SYSTEM

VECTOR FROM / TO

X 0 Y 0 Z 1

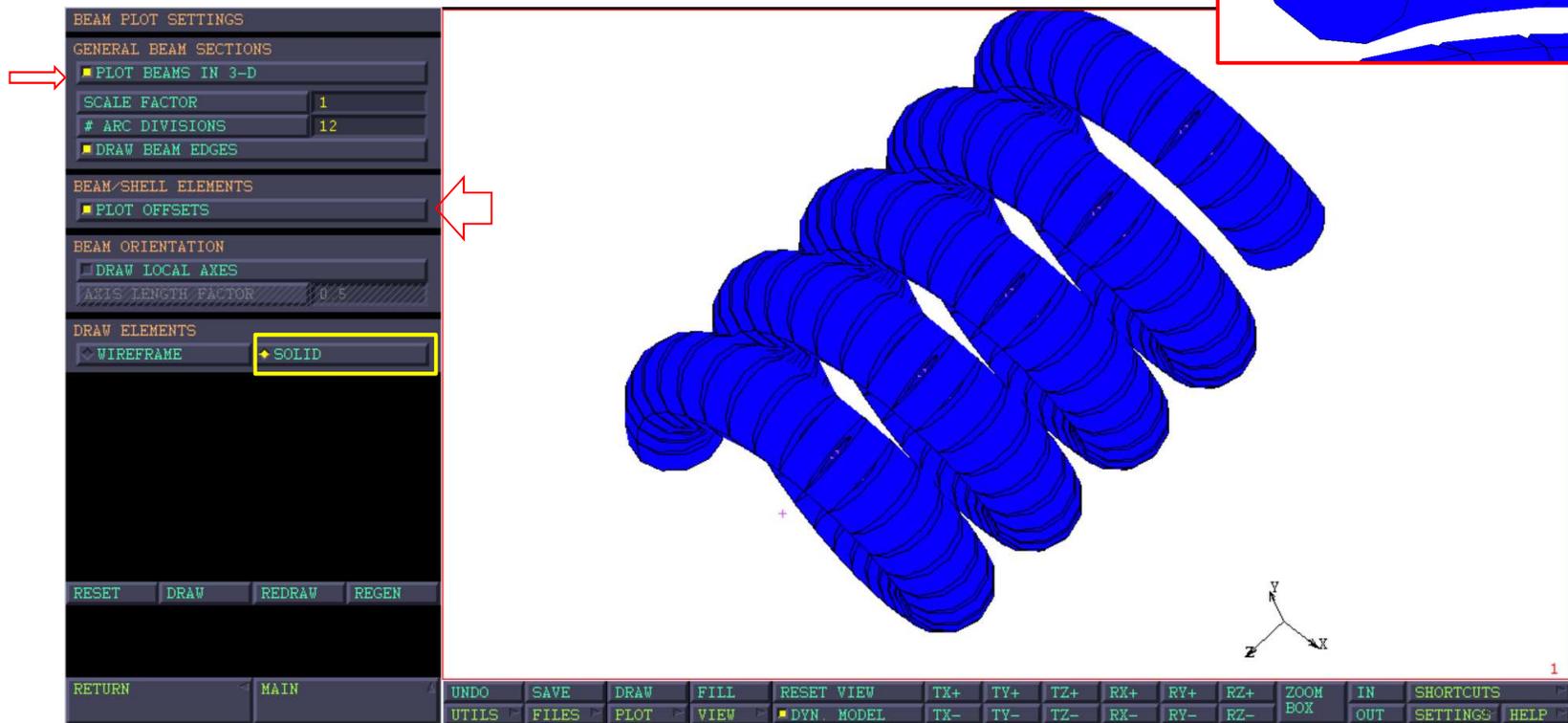
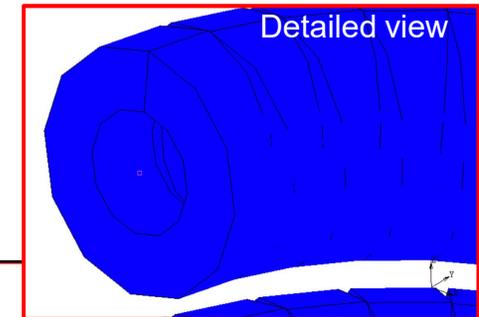
BEAM CONTACT BEAM OFFSETS

CLEAR OK

The plot of the coil in 3D might be checked from the PLOT SETTINGS BEAM menu.

# Geometric properties

## Hollow circular cross-section



Deflag the PLOT BEAMS IN 3D menu, to return to the beam model visualisation.

# Material properties

## Isotropic and homogeneous

The modal analysis requires the evaluation of the mass matrix related to the component and therefore the density of the material must be set.

The screenshot shows the 'MATERIAL PROPERTIES' dialog box in a CAD application. The 'ANALYSIS CLASS' is set to 'STRUCTURAL'. The 'NAME' is 'titanium' and the 'TYPE' is 'standard'. The 'GENERAL PROPERTIES' dialog is open, showing 'MASS DENSITY' set to  $4.7e-09$ . The 'DESIGN SENSITIVITY/OPTIMIZATION' section shows 'COST / VOLUME' and 'COST / MASS' both set to 0. The 'OK' button is highlighted in yellow. A 3D model of a spring is visible in the background. The bottom status bar shows various utility buttons like UNDO, SAVE, DRAW, FILL, RESET VIEW, TX+, TY+, TZ+, RX+, RY+, RZ+, ZOOM BOX, IN, SHORTCUTS, UTILS, FILES, PLOT, VIEW, DYN. MODEL, TX-, TY-, TZ-, RX-, RY-, RZ-, OUT, SETTINGS, and HELP.

GENERAL  
MASS DENSITY  
4.7e-9  
Press Enter  
OK

# Material properties

## Isotropic and homogeneous

Then, the definition of the elastic constant of the titanium allows the stiffness matrix to be evaluated.

The general and the structural properties of the material must be applied to the mesh elements.

The screenshot displays the material definition interface. On the left, the 'MATERIAL PROPERTIES' panel shows the material name 'titanium' and its classification as 'STRUCTURAL'. The 'DATA CATEGORIES' section includes 'GENERAL' and 'STRUCTURAL'. The 'ELEMENTS' section shows 'ADD' and 'REM' buttons, with '144' elements listed. The right panel, 'STRUCTURAL PROPERTIES', shows the material type as 'ELASTIC-PLASTIC ISOTROPIC' and the elastic constants: 'YOUNG'S MODULUS' (110000) and 'POISSON'S RATIO' (0.3). A 'RESET' button is visible. The bottom status bar contains various utility buttons.

# LINKS

## RBE2

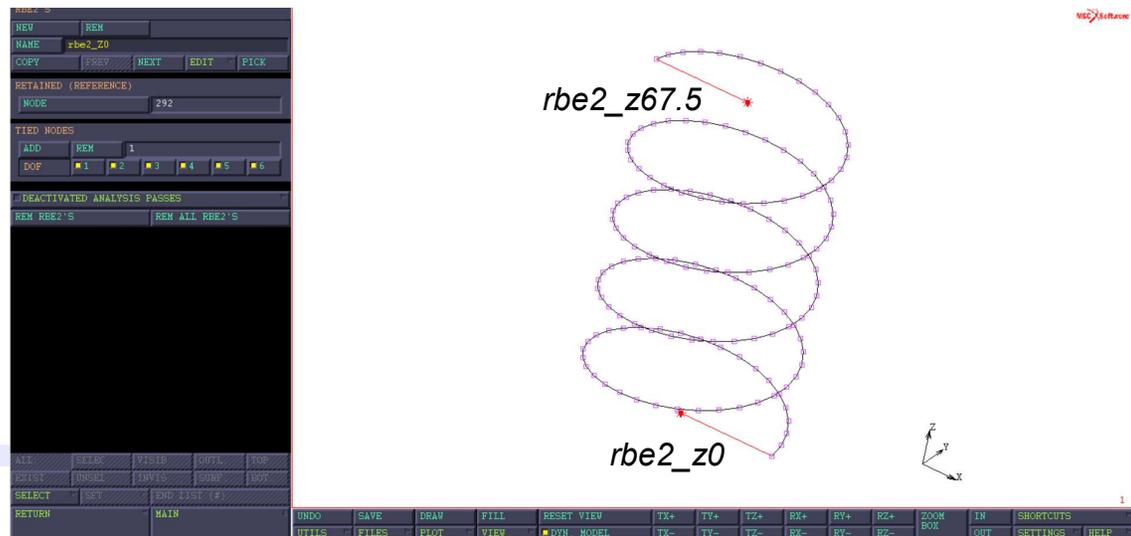
```

-----
| I create two nodes at the coil axis at Z=0
| and at the other free extremity Z = 67.5
|-----
*add_nodes
0
0
0
0
0
4.5*15
|-----
|   RBE2 at Z=0
|-----
*new_rbe2
*rbe2_name rbe2_z0
*rbe2_ret_node
  292
*add_rbe2_tied_nodes
  1
# | End of List
*rbe2_tied_dof 1
*rbe2_tied_dof 2
*rbe2_tied_dof 3
*rbe2_tied_dof 4
*rbe2_tied_dof 5
*rbe2_tied_dof 6
|-----
|   RBE2 at Z=67.5
|-----
*new_rbe2
*rbe2_name rbe2_z67.5
  291
*add_rbe2_tied_nodes
  290
# | End of List
*rbe2_tied_dof 1
*rbe2_tied_dof 2
*rbe2_tied_dof 3
*rbe2_tied_dof 4
*rbe2_tied_dof 5
*rbe2_tied_dof 6
|-----

```

To load and constrain the coil, two RBE2 are defined located at the lower and the upper extremities of the coil, named *rbe2\_z0* and *rbe2\_z67.5*, respectively.

The retained node is located at the coil axial and the tied node is considered at the coil vertex in both RBE2 cases.



# Agenda

Goal and nomenclature

Model setup

**Modal Analysis loadcases varying the BCs:**

- **Free-free**
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- Considering damping effect

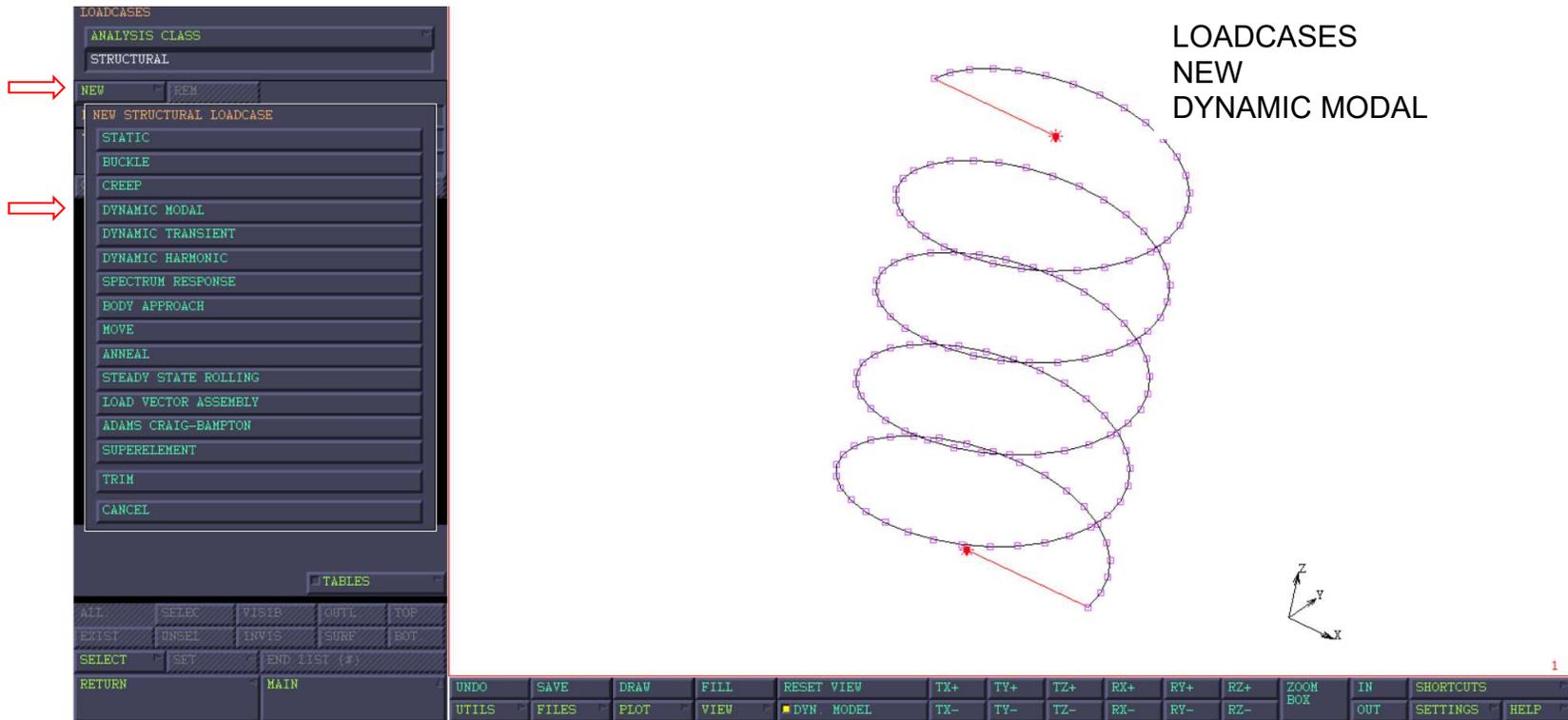
References

# Loadcases

## Modal analysis

The modal analysis is a dynamic analysis, therefore the loading conditions to be adopted are in:

LOADCASES  
NEW  
DYNAMIC MODAL



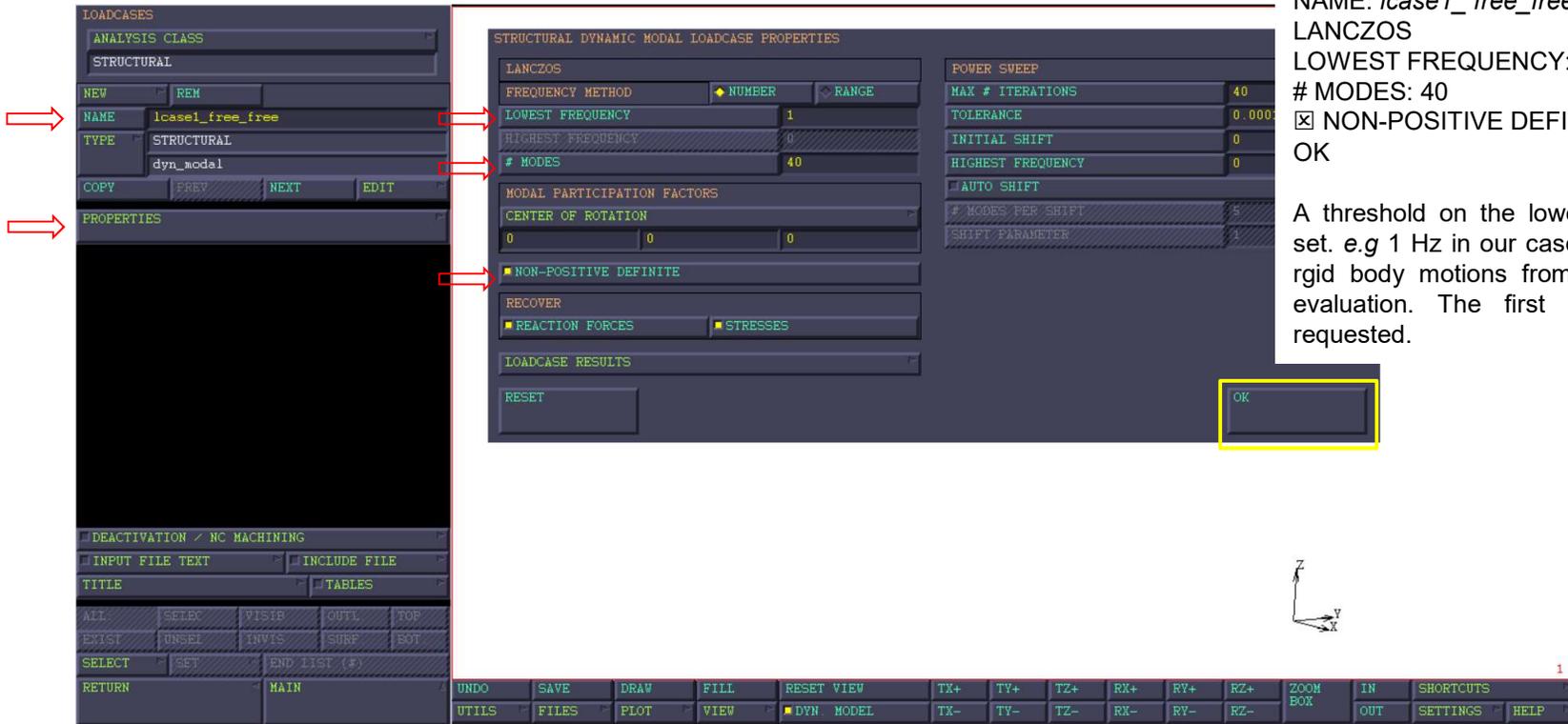
# Loadcases

## Modal analysis: free-free

To collect the first natural modes neglecting the six rigid body motions due to the absence of any BCs (free-free modal analysis, the model must be set as follows:

```
LOADCASES
NEW
DYNAMIC MODAL
NAME: lcase1_free_free
LANCZOS
LOWEST FREQUENCY: 1
# MODES: 40
 NON-POSITIVE DEFINITE
OK
```

A threshold on the lowest frequency is set. e.g 1 Hz in our case, to remove the rigid body motions from the numerical evaluation. The first 40 modes are requested.



Remember that the modal response of the structure is influenced by the BCs acting on the component.

# Jobs

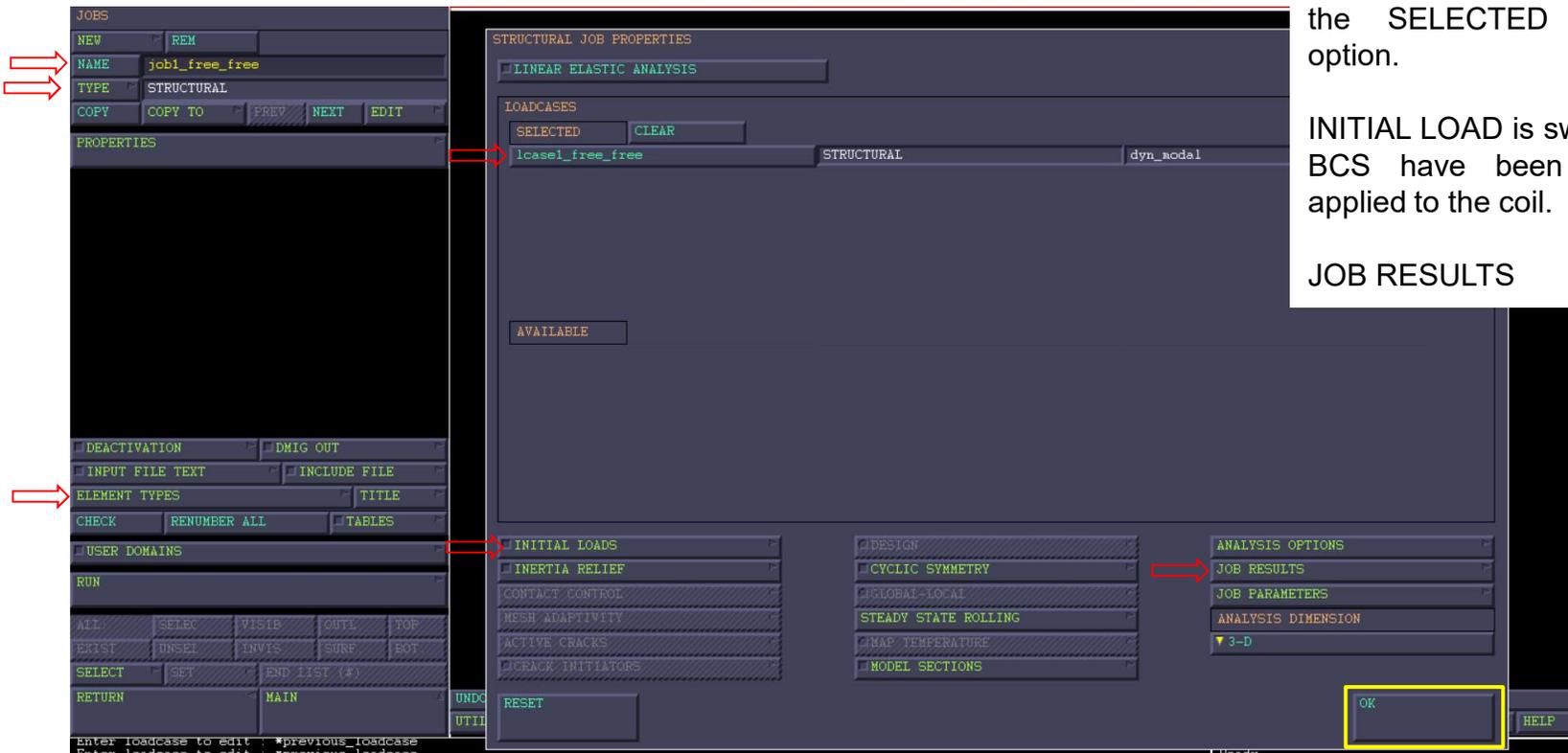
## Job results

JOBS  
NEW  
TYPE: STRUCTURAL  
NAME: job1\_free\_free  
PROPERTIES

The *lcase1\_free\_free* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

INITIAL LOAD is switched off, any BCS have been defined and applied to the coil.

JOB RESULTS



# Jobs

## Element Type

JOB  
ELEMENT TYPE  
3D  
TRUSS/BEAM  
CLOSED SECTION BEAM  
14  
OK

For details, see the HELP guide, the positions of the 16 numerical integration point proper of this elementis reported.

The screenshot shows the 'Marc Mentat 3-D Line Structural Element Types' dialog box. The 'ANALYSIS CLASS' is set to 'STRUCTURAL'. The 'LINE' table lists various element types, with '14' highlighted in a yellow box. To the right, a diagram of a circular cross-section shows 16 integration points numbered 1 through 16, with a radius and thickness indicated. The 'HELP' button in the bottom right corner of the dialog box is also highlighted in yellow.

	LINE	
TRUSS	9	64
THIN ELASTIC BEAM	52	
THICK ELASTIC BEAM	38	
CLOSED SECTION BEAM	14	
CLOSED SECTION BEAM with VAR. AXIAL STRAIN	25	
CLOSED SECTION BEAM	78	76
OPEN SECTION BEAM with WARPING	79	77
CABLE	51	
ELBOW	31	

# Jobs

## Results: Beam Element

The 1D beam stress are requested and evaluated on the element layers (16 integration point), as shown in the help guide.

The screenshot shows the 'JOB RESULTS' dialog box with the following sections:

- POST FILE:** BINARY, DEFAULT STYLE, INCREMENT FREQUENCY: 1
- OUTPUT FILE:** REPAIR VERIFICATION, FLOWLINES, TRACKING, STATUS FILE, FORCE BALANCE
- ADDITIONAL CONTACT FILES:** (empty)
- SELECTED ELEMENT QUANTITIES:** CLEAR button, LAYERS dropdown (ALL), and a list of quantities: Beam Orientation Vector, Beam Axial Force, Beam Bending Moment Local X, Beam Bending Moment Local Y, Beam Shear Force Local X, Beam Shear Force Local Y, and Beam Torsional Moment.
- AVAILABLE ELEMENT TENSORS:** Elastic Left Cauchy-Green Deform, Elastic Part Left Cauchy-Green D, Real Harmonic Stress, Imag Harmonic Stress, Real Harmonic Shell Curvature, and Imag Harmonic Shell Curvature.
- AVAILABLE ELEMENT SCALARS:** Beam Bending Moment Local Y, Beam Shear Force Local X, Beam Shear Force Local Y, Beam Torsional Moment, Beam Bimoment, and Gasket Pressure.
- AVAILABLE NODAL QUANTITIES:** Stress Intensity Mode III, VCCT Energy Release, VCCT Energy Release I, VCCT Energy Release II, VCCT Energy Release III, and VCCT Failure Index.
- ELEMENT RESULTS:** ALL POINTS, CENTROID
- SELECTED NODAL QUANTITIES:** CLEAR button, DEFAULT, CUSTOM, and a list: Displacement, Rotation, External Force, External Moment, and Reaction Force.
- CONTACT GLUE FORCES:** INCLUDE, EXCLUDE
- ITERATIVE RESULTS:** OFF
- OK** button (highlighted in yellow)

```

-----
|          JOBS START          |
|-----|
*add_job_loadcases
lcase1_free_free
*job_name job1
*post_eq_all_layers stress
*post_eq_max_min_layers von_mises
*job_option nod_quantities:manual
*add_post_nodal_quantity Displacement
*add_post_nodal_quantity Rotation
*add_post_nodal_quantity Ext_Force
*add_post_nodal_quantity Ext_Moment
*add_post_nodal_quantity Rea_Force
*add_post_nodal_quantity Rea_Moment
*add_post_nodal_quantity Ty_Force
*add_post_nodal_quantity Ty_Moment
|-----|
| Components of stress occurring in the |
| coil modelled by 1D elements         |
|-----|
*add_post_var bm_orient
*add_post_var bm_axi_for
*add_post_var bm_bnd_mom_x
*add_post_var bm_bnd_mom_y
*add_post_var bm_shr_for_x
*add_post_var bm_shr_for_y
*add_post_var bm_tor_mom
*post_eq_all_layers bm_orient
*post_eq_all_layers bm_axi_for
*post_eq_all_layers bm_bnd_mom_x
*post_eq_all_layers bm_bnd_mom_y
*post_eq_all_layers bm_orient
*post_eq_all_layers bm_shr_for_x
*post_eq_all_layers bm_shr_for_y
*post_eq_all_layers bm_tor_mom
*element_type 14
|-----|
|          JOBS FINISH          |
|-----|

```

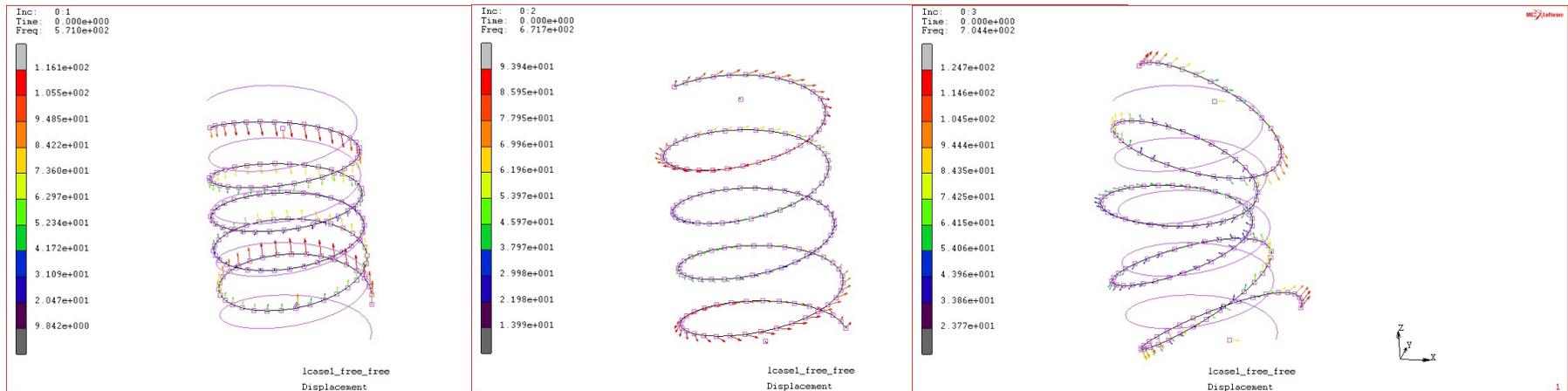
Now, check, save and run the model!!



# Results

## Natural frequencies of the coil (#3 modes)

Check the deformation mode of the model in comparison with the undeformed condition.  
The scaled factor applied to compare the deformation is set equal to 0.1.  
In addition, the displacement field is plotted by vectors from the RESULT MORE menu.



# 1 mode  
Axial compressive mode  
Freq: 571.0 Hz

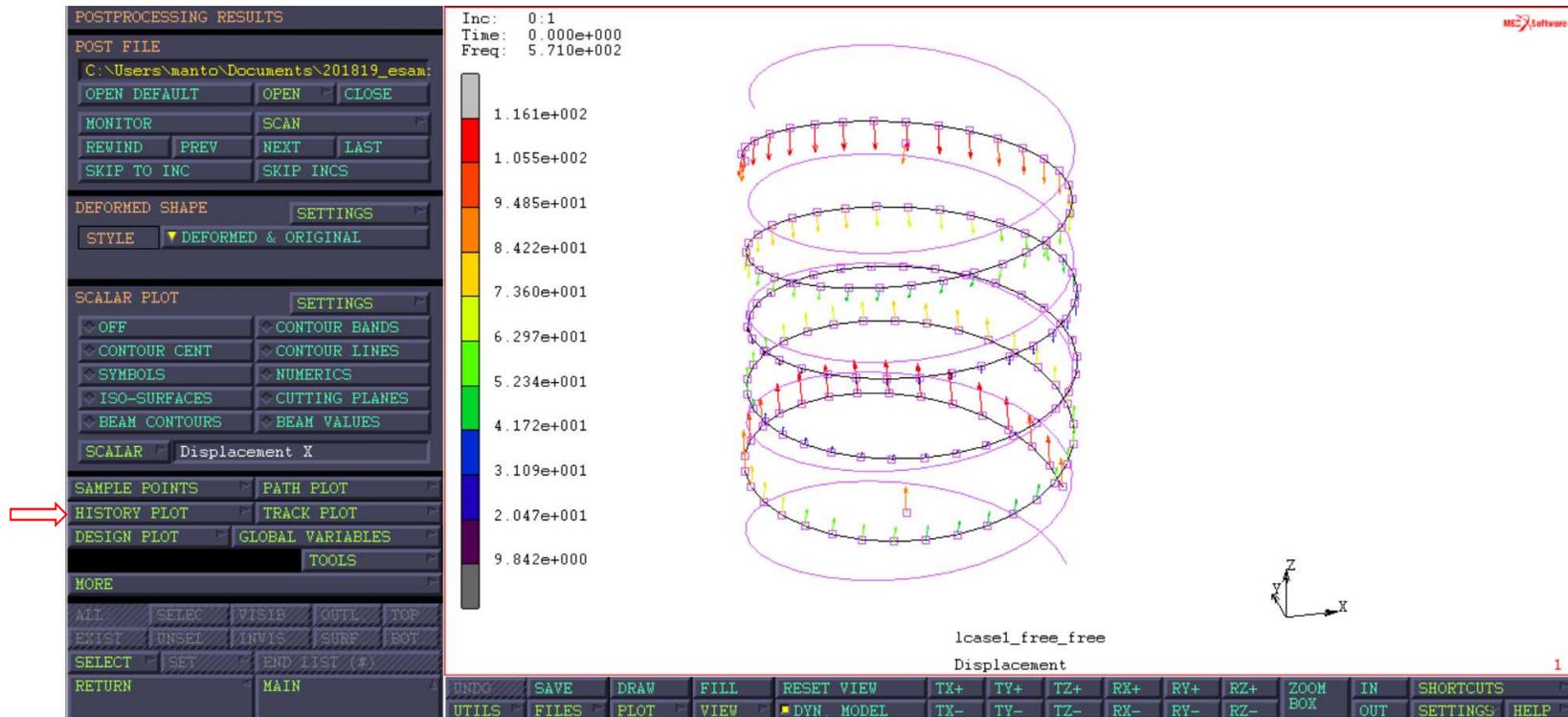
# 2 mode  
Circumferential Expansion mode  
Freq: 671.7Hz

# 3 mode  
Lateral bending mode  
Freq: 704.4 Hz

# Results

## Natural frequencies: Hystory plot

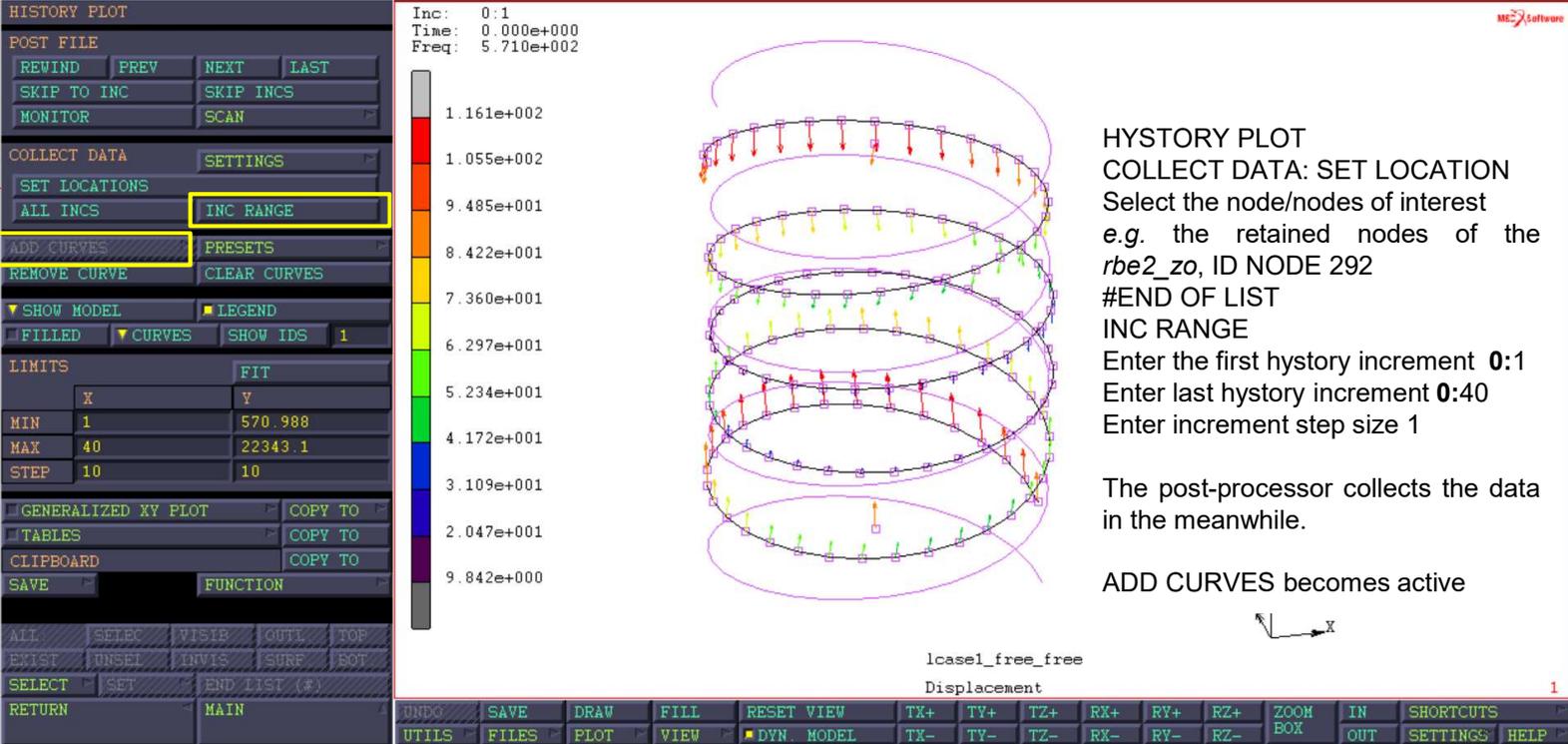
The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



The screenshot displays the Hystory Plot software interface. On the left is a control panel with various options. A red arrow points to the 'SET LOCATIONS' option under 'COLLECT DATA'. A yellow box highlights the 'INC RANGE' option. The central area shows a 3D model of a coil with displacement vectors and a color scale for displacement. The scale ranges from 9.842e+000 to 1.161e+002. The bottom of the interface shows a command prompt window with the following text:

```
Command: Hystory_Plot
Enter first history increment : 0:1
Enter last history increment : 0:40
Enter increment step size : 1
```

HYSTORY PLOT  
COLLECT DATA: SET LOCATION  
Select the node/nodes of interest  
e.g. the retained nodes of the  
*rbe2\_zo*, ID NODE 292  
#END OF LIST  
INC RANGE  
Enter the first hystory increment 0:1  
Enter last hystory increment 0:40  
Enter increment step size 1

The post-processor collects the data  
in the meanwhile.

ADD CURVES becomes active

# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.

HISTORY PLOT CURVES

ADD CURVES

SINGLE LOCATION ALL LOCATIONS

LOC1 vs LOC2 GLOBAL

REMOVE CURVE CLEAR CURVES FIT

LOCATIONS

NODE 291

GLOBAL VARIABLES

Increment

Sub Increment

Time

Frequency

CONTACT BODY VARIABLES

VARIABLES AT LOCATIONS

Displacement X

Displacement Y

Displacement Z

Rotation X

RETURN MAIN

HYSTORY PLOT

COLLECT DATA: SET LOCATION

Select the node/nodes of interest  
e.g. the retained nodes of the *rbe2\_zo*, ID NODE 292

#END OF LIST

INC RANGE

Enter the first hystory increment 0:1

Enter last hystory increment 0:40

Enter increment step size 1

The post-processor collects the data in the meanwhile.

ADD CURVES becomes active

ADD CURVES:

ALL LOCATIONS

GLOBAL VARIABLES:

SUB INCREMENT (The first: x-axis)

FREQUENCY (The second y-axis)

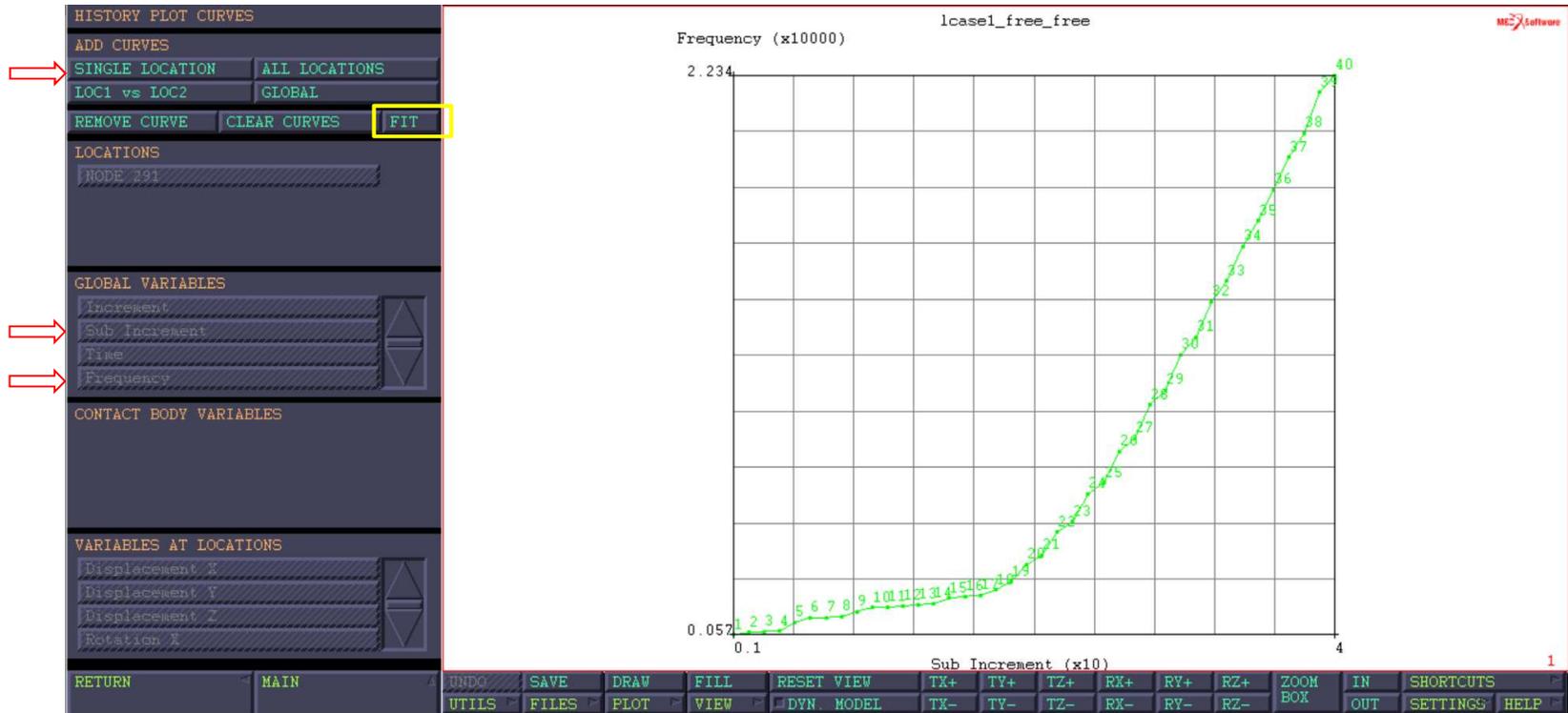
FIT

1

# Results

## Natural frequencies: Hystory plot

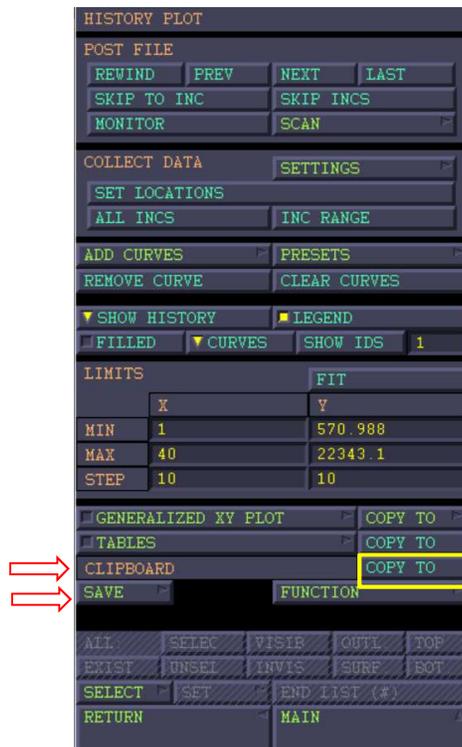
The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



```
HISTORY PLOT
job1_free_free
```

```
Curve 1
X : Sub Increment
Y : Frequency
```

X	Y
1.000000e+000	5.709879e+002
2.000000e+000	6.716981e+002
3.000000e+000	7.044292e+002
4.000000e+000	7.165169e+002
5.000000e+000	1.056302e+003
6.000000e+000	1.245774e+003
7.000000e+000	1.248814e+003
8.000000e+000	1.278027e+003
9.000000e+000	1.482664e+003
1.000000e+001	1.636913e+003
1.100000e+001	1.645826e+003
1.200000e+001	1.693099e+003
1.300000e+001	1.747204e+003
1.400000e+001	1.790561e+003

**CLIPBOARD:**  
copy and paste the data from MARC to the Excel file only for Windows.

**SAVE:**  
Save the data in a text file in a defined directory, The .txt file formatting is presented beside.

# Agenda

Goal and nomenclature

Model setup

**Modal Analysis loadcases varying the BCs:**

- Free-free
- **Fixed-fixed**
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

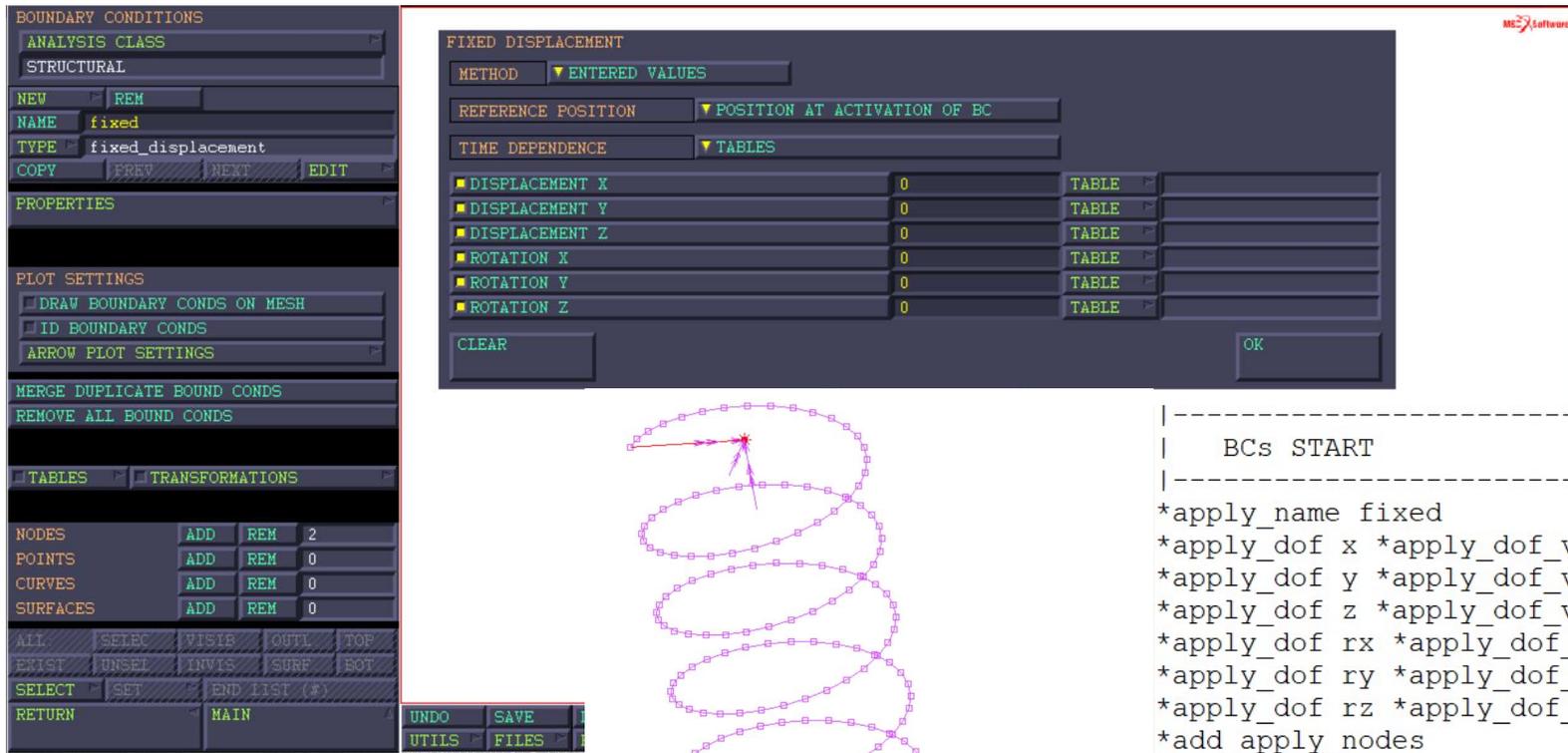
- Neglecting damping effect
- Considering damping effect

References

# BCs

## Fixed at both coil extremities

Moving from a free-free modal analysis to a fixed-fixed modal analysis, the natural frequencies of the coil are evaluated. The natural frequency involves the modes proper of the wire preventing the movement of the extremity of the component.



The image displays the ANSYS software interface. On the left, the 'BOUNDARY CONDITIONS' panel shows 'ANALYSIS CLASS' set to 'STRUCTURAL' and 'TYPE' set to 'fixed\_displacement'. The 'FIXED DISPLACEMENT' dialog box is open, showing 'METHOD' as 'ENTERED VALUES' and 'REFERENCE POSITION' as 'POSITION AT ACTIVATION OF BC'. The 'TIME DEPENDENCE' section is set to 'TABLES'. The dialog box lists displacement and rotation values for X, Y, and Z axes, all set to 0. Below the dialog box, a 3D model of a coil is shown with red arrows indicating fixed boundary conditions at both ends. A coordinate system (x, y, z) is visible at the bottom right of the model.

```
-----  
| BCs START  
|-----  
*apply_name fixed  
*apply_dof x *apply_dof_value x  
*apply_dof y *apply_dof_value y  
*apply_dof z *apply_dof_value z  
*apply_dof rx *apply_dof_value rx  
*apply_dof ry *apply_dof_value ry  
*apply_dof rz *apply_dof_value rz  
*add_apply_nodes  
291  
292  
# | End of List  
|-----
```



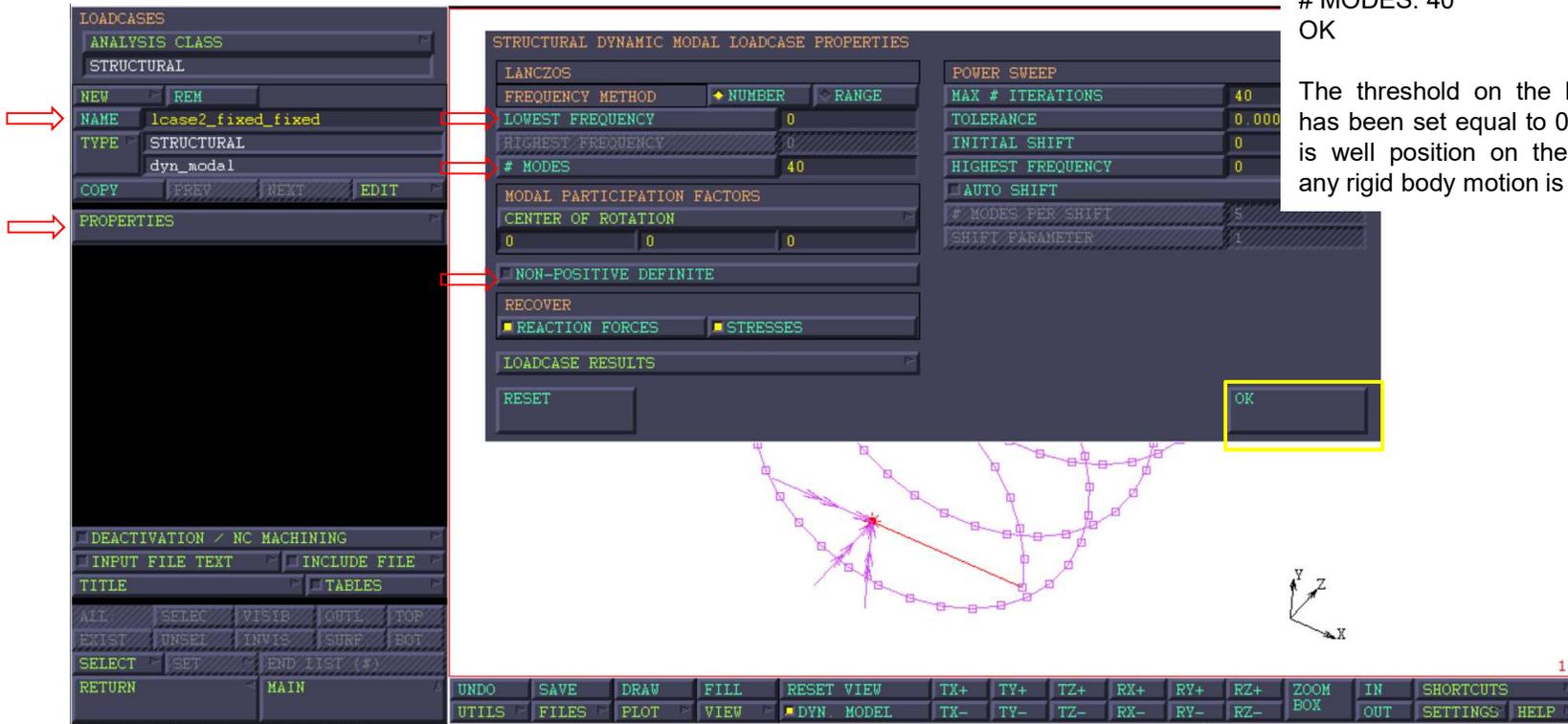
# Loadcases

## Modal analysis: fixed-fixed

The model loadcase must be set as follows:

```
LOADCASES
NEW
DYNAMIC MODAL
NAME: lcase2_fixed_fixed
LANCZOS
LOWEST FREQUENCY: 0
# MODES: 40
OK
```

The threshold on the lowest frequency has been set equal to 0. The component is well position on the modelling area, any rigid body motion is prevented.



Remember that the modal response of the structure is influenced by the BCs acting on the component.

# Jobs

## Job results

JOB  
NEW  
TYPE: STRUCTURAL  
NAME: job2\_fixed\_fixed  
PROPERTIES

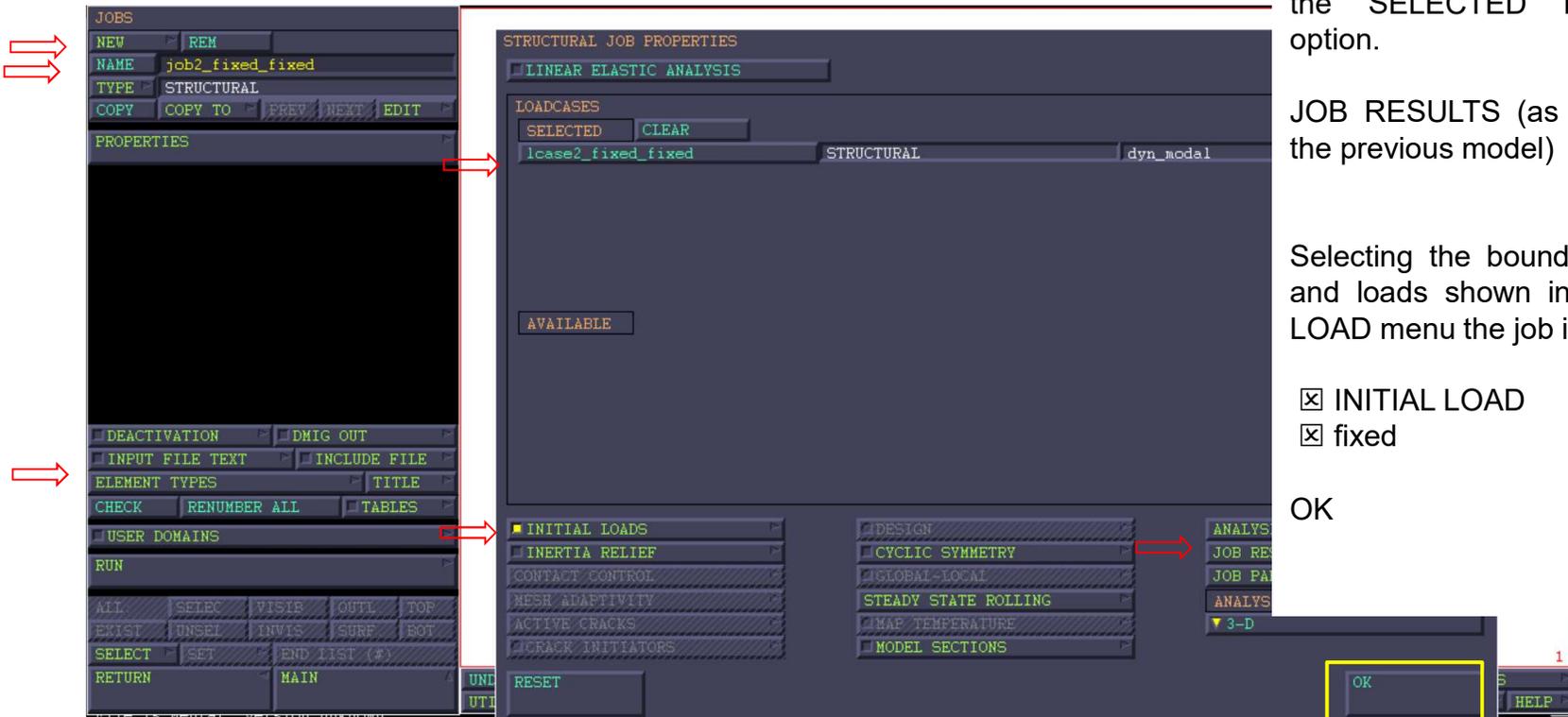
The *lcase2\_fixed\_fixed* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

JOB RESULTS (as discussed in the previous model)

Selecting the boundary condition and loads shown in the INITIAL LOAD menu the job is defined.

- INITIAL LOAD
- fixed

OK



# Jobs

## Job results

JOBS  
 NEW  
 TYPE: STRUCTURAL  
 NAME: job2\_fixed\_fixed  
 PROPERTIES

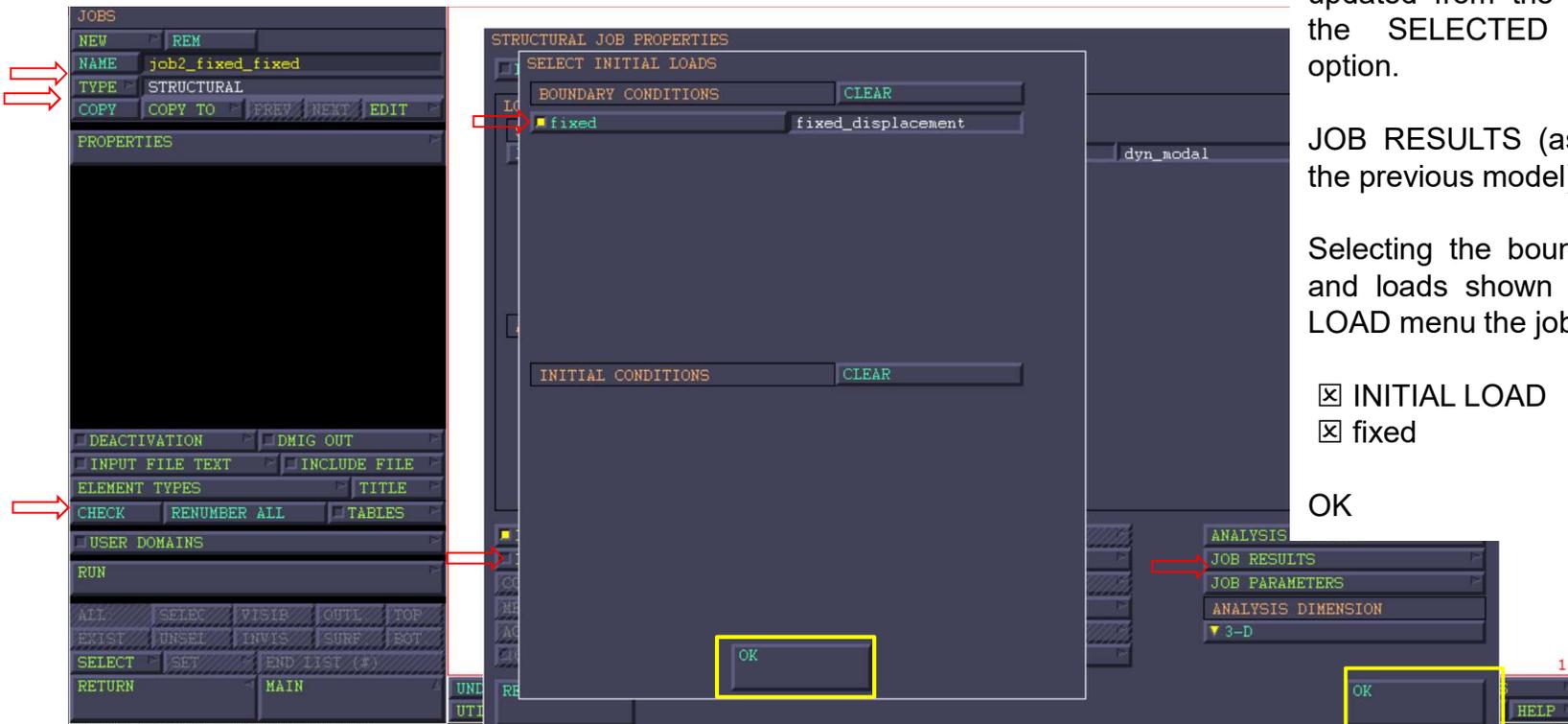
The *lcase2\_fixed\_fixed* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

JOB RESULTS (as discussed in the previous model)

Selecting the boundary condition and loads shown in the INITIAL LOAD menu the job is defined.

- INITIAL LOAD
- fixed

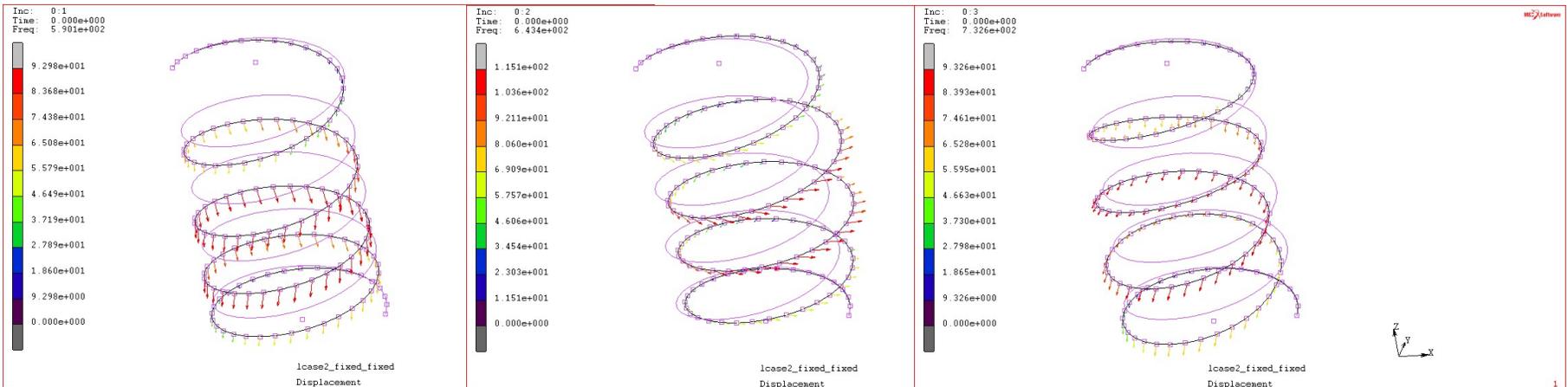
OK



# Results

## Natural frequencies of the coil (#3 modes)

Check the deformation mode of the model in comparison with the undeformed condition.  
The scaled factor applied to compare the deformation is set equal to 0.1.  
In addition, the displacement field is plotted by vectors from the RESULT MORE menu.



# 1 mode  
Axial compressive mode  
Freq: 590.1 Hz

# 2 mode  
Circumferential Expansion - lateral  
bending mode  
Freq: 643.4Hz

# 3 mode  
Mixed mode  
Freq: 732.6 Hz

# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



HISTORY PLOT  
job2\_fixed\_fixed

Curve 1  
X : Sub Increment  
Y : Frequency

X	Y
1.000000e+000	5.901064e+002
2.000000e+000	6.434000e+002
3.000000e+000	7.325642e+002
4.000000e+000	7.561287e+002
5.000000e+000	1.112751e+003
6.000000e+000	1.163618e+003
7.000000e+000	1.262487e+003
8.000000e+000	1.324692e+003
9.000000e+000	1.484989e+003
1.000000e+001	1.626085e+003
1.100000e+001	1.647429e+003
1.200000e+001	1.692789e+003
1.300000e+001	1.793034e+003
1.400000e+001	1.795375e+003

CLIPBOARD:  
copy and paste directly  
the data form MARC to  
the Excel file only for  
Windows.

SAVE:  
Save the data in a text  
file in a defined directory,  
The .txt file formatting is  
presented beside.

# Agenda

Goal and nomenclature

Model setup

**Modal Analysis loadcases varying the BCs:**

- Free-free
- Fixed-fixed
- **Fixed-axial rotation free**
- Fixed-displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- Considering damping effect

References

# BCs

## Upper extremity fixed – lower extremity rot<sub>z</sub> free

Moving from a free-free modal analysis to a fixed-fixed modal analysis, the natural frequencies of the coil are evaluated. The natural frequency involves the modes proper of the wire preventing the movement of the extremity of the component.

The screenshot displays the software interface for setting boundary conditions. On the left, the 'BOUNDARY CONDITIONS' panel shows the 'ANALYSIS CLASS' set to 'STRUCTURAL'. The 'FIXED DISPLACEMENT' panel on the right shows the 'METHOD' set to 'ENTERED VALUES' and the 'REFERENCE POSITION' set to 'POSITION AT ACTIVATION OF BC'. The 'TIME DEPENDENCE' is set to 'TABLES'. The displacement and rotation values are all set to 0.

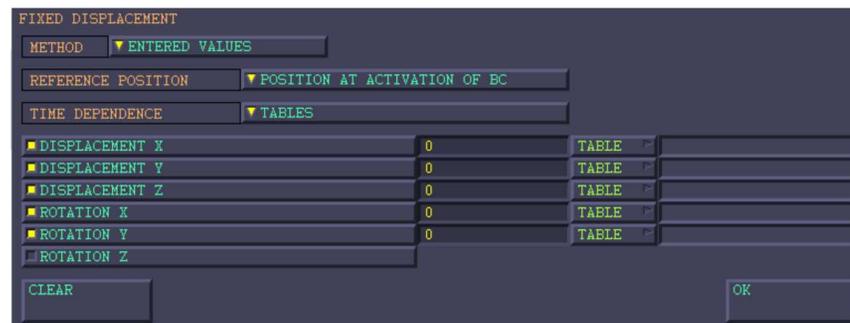
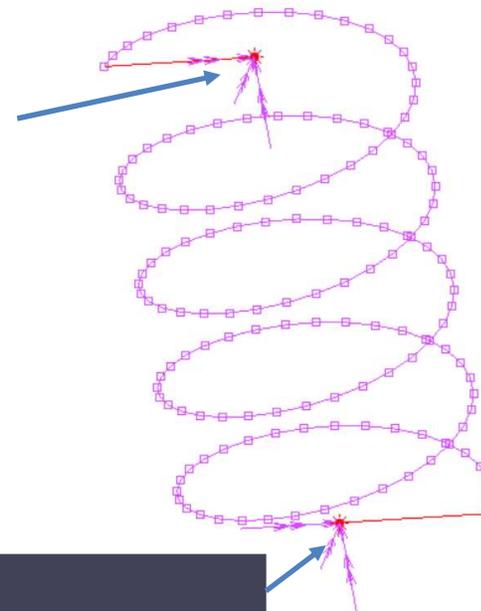
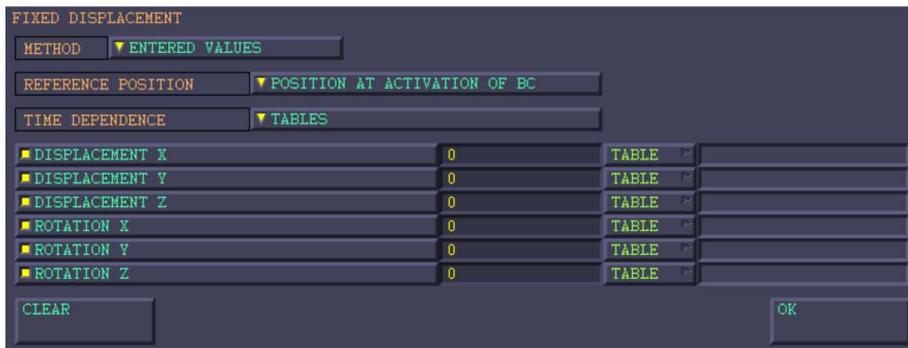
PROPERTY	VALUE	TABLE
DISPLACEMENT X	0	TABLE
DISPLACEMENT Y	0	TABLE
DISPLACEMENT Z	0	TABLE
ROTATION X	0	TABLE
ROTATION Y	0	TABLE
ROTATION Z	0	TABLE

The 3D model shows a coil with two red dots indicating fixed boundary conditions at the top and bottom. A coordinate system (X, Y, Z) is shown at the bottom right of the model.

# BCs

## Upper extremity fixed – lower extremity rot<sub>z</sub> free

The natural frequencies of the coil are evaluated, considering the upper extremity of the coil as fixed, and the lower one as free to rotate along Z direction. Therefore, the BCs are named *fixed* and *rotz\_free*.



# Loadcase

Upper extremity fixed – lower extremity rot<sub>z</sub> free

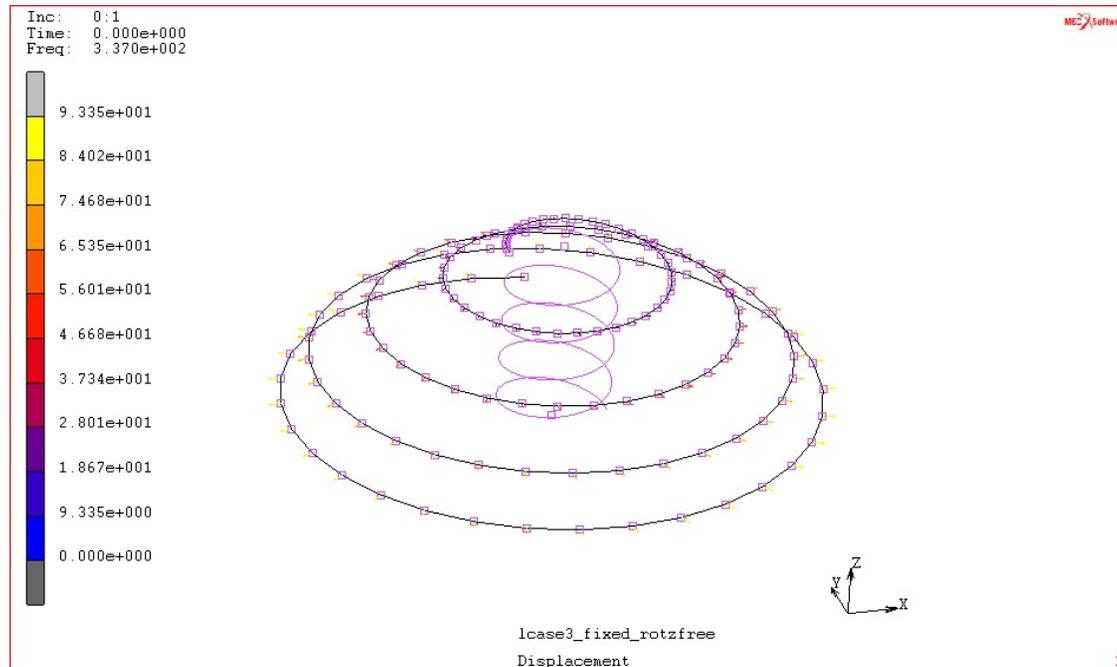
Remember to update the INITIAL LOAD condition at the JOBS menu, then submit the model to the evaluation.



# Results

## Natural frequencies of the coil (#1 mode)

Check the deformation mode of the model in comparison with the undeformed condition.  
The scaled factor applied to compare the deformation is set equal to 1.0  
In addition, the displacement field is plot by vectors from the RESULT MORE menu.



# 1 mode  
Circumferential Expansion of the wire  
Freq: 337.0 Hz

# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



HISTORY PLOT  
job3\_fixed\_rotzfree

Curve 1  
X : Sub Increment  
Y : Frequency

X	Y
1.000000e+000	3.369980e+002
2.000000e+000	5.921451e+002
3.000000e+000	7.263797e+002
4.000000e+000	7.325867e+002
5.000000e+000	9.684847e+002
6.000000e+000	1.131105e+003
7.000000e+000	1.217419e+003
8.000000e+000	1.262912e+003
9.000000e+000	1.460717e+003
1.000000e+001	1.535286e+003
1.100000e+001	1.638096e+003
1.200000e+001	1.647432e+003
1.300000e+001	1.762460e+003
1.400000e+001	1.793168e+003

CLIPBOARD:  
copy and paste the data from MARC to the Excel file only for Windows.

SAVE:  
Save the data in a text file in a defined directory, The .txt file formatting is presented beside.

# Agenda

Goal and nomenclature

Model setup

**Modal Analysis loadcases varying the BCs:**

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- **Fixed-axial displacement and rotation free**

Harmonic loadcases:

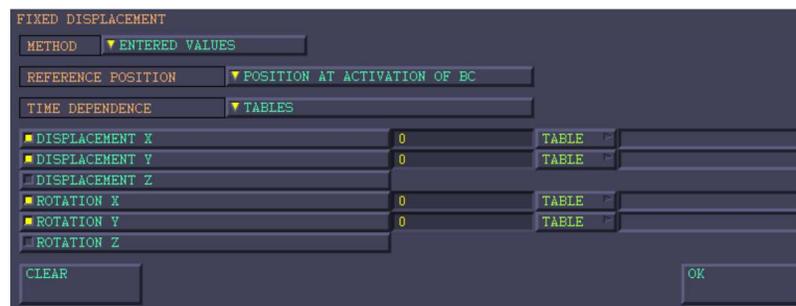
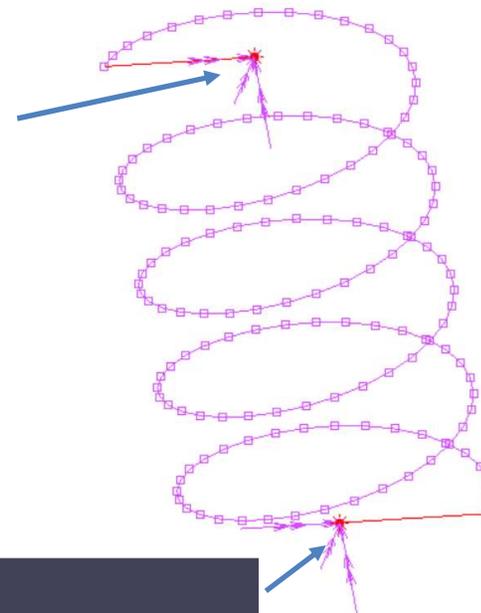
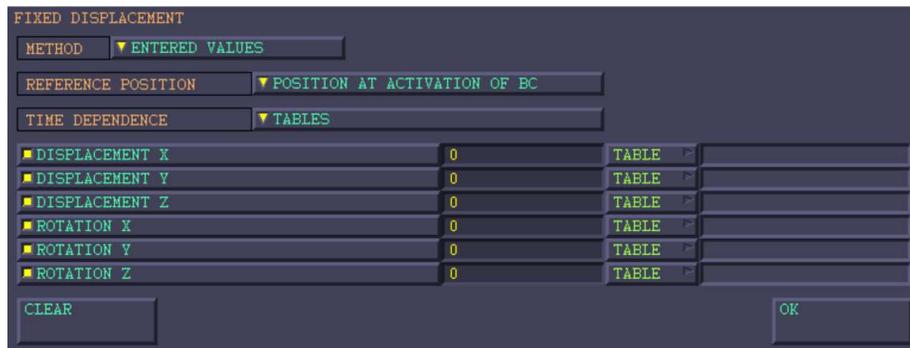
- Neglecting damping effect
- Considering damping effect

References

# BCs

Upper extremity fixed – lower extremity  $displ_z$  and  $rot_z$  free

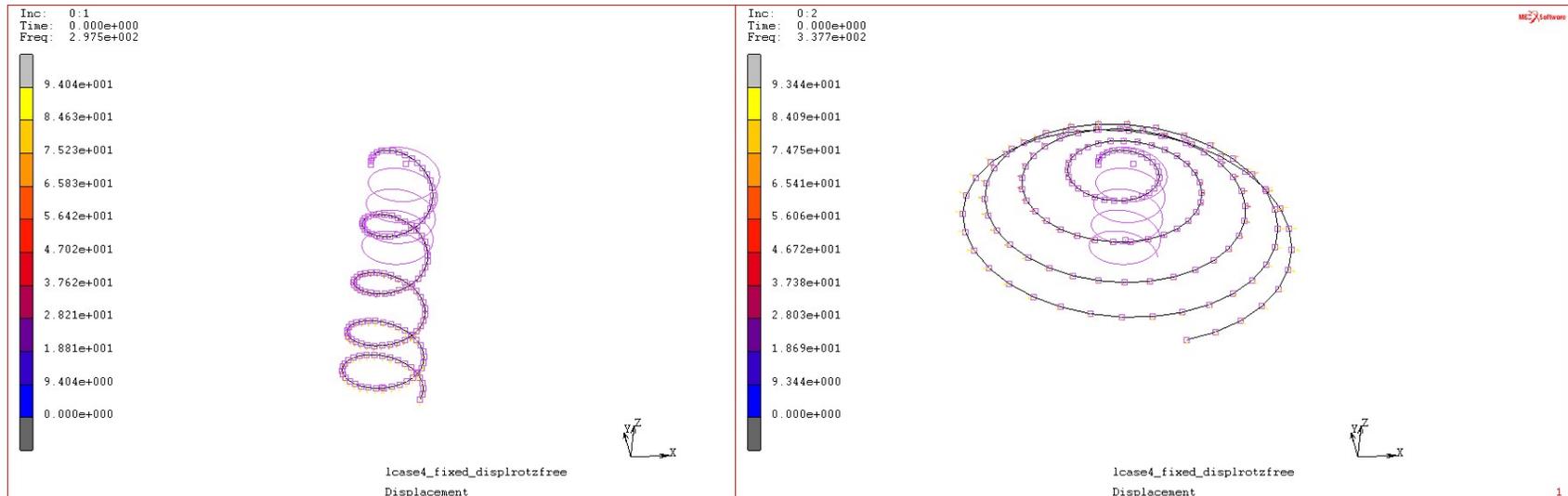
The natural frequencies of the coil are evaluated, considering the upper extremity of the coil as fixed, and the lower one as free to move and to rotate along Z direction. Therefore, the BCs are two named *fixed* and *displrotz\_free*.



# Results

## Natural frequencies of the coil (#1 and #2 mode)

Check the deformation mode of the model in comparison with the undeformed condition.  
The scaled factor applied to compare the deformation is set equal to 1.0  
In addition, the displacement field is plotted by vectors from the RESULT MORE menu.



# 1 mode  
Axial displacement  
Freq: 297.5 Hz

# 2 mode  
Circumferential Expansion of the wire  
Freq: 337.7 Hz

# Results

## Natural frequencies: Hystory plot

The automatic collection of natural frequencies of the coil (#40 modes) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files.



```
HISTORY PLOT
lcase4_fixed_displrotzfree
```

```
Curve 1
X : Sub Increment
Y : Frequency
```

X	Y
1.000000e+000	2.975393e+002
2.000000e+000	3.377146e+002
3.000000e+000	7.263588e+002
4.000000e+000	7.320267e+002
5.000000e+000	8.665751e+002
6.000000e+000	9.744045e+002
7.000000e+000	1.216291e+003
8.000000e+000	1.262441e+003
9.000000e+000	1.328036e+003
1.000000e+001	1.504377e+003
1.100000e+001	1.625996e+003
1.200000e+001	1.640287e+003
1.300000e+001	1.669783e+003
1.400000e+001	1.768155e+003

CLIPBOARD:  
copy and paste directly  
the data form MARC to  
the Excel file only for  
Windows.

SAVE:  
Save the data in a text  
file in a defined directory,  
The .txt file formatting is  
presented beside.

# Agenda

Goal and nomenclature

Model setup

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

**Harmonic loadcases:**

- **Neglecting damping effect**
- Considering damping effect

References

# Frequency response

## Considerations

When a system is being excited by known oscillatory frequencies, it may not be feasible to design its natural frequencies out of this operating range. In cases of this nature, evaluating the system in the presence of this enforced vibration proves necessary. When the excitation does not change with time, the solution is a steady state response at the operating frequency of interest. This is known as *frequency response analysis*. The relevant results of this analysis are typically displacements, velocities, and accelerations of the system, which can be used to calculate forces and stress in the structure.

The presence of an harmonic force ( $F$ ) or displacement ( $u$ ) is defined as follows:  $u = u_0 \cos(\omega t + \varphi)$  or  $F = F_0 \cos(\omega t + \varphi)$

Where

$\omega$  is the angular frequency

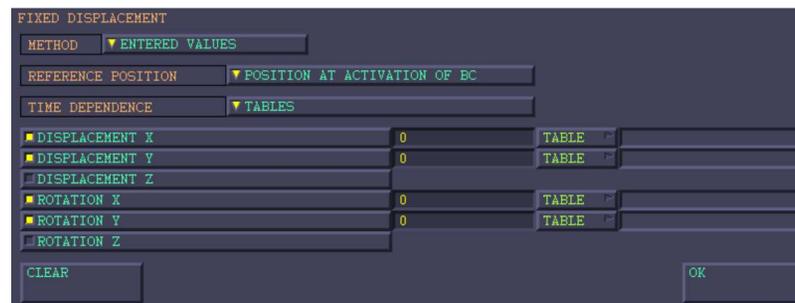
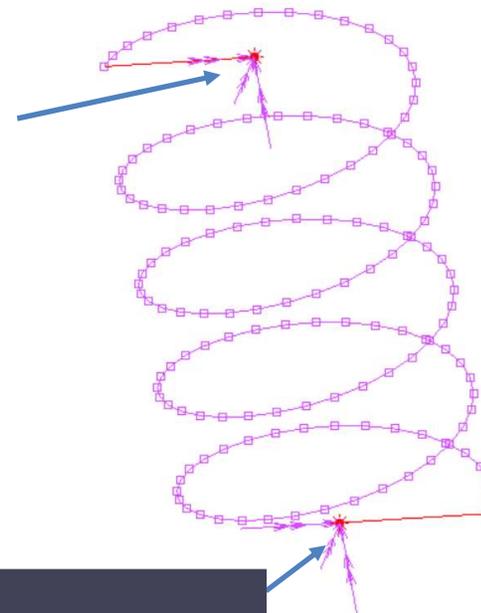
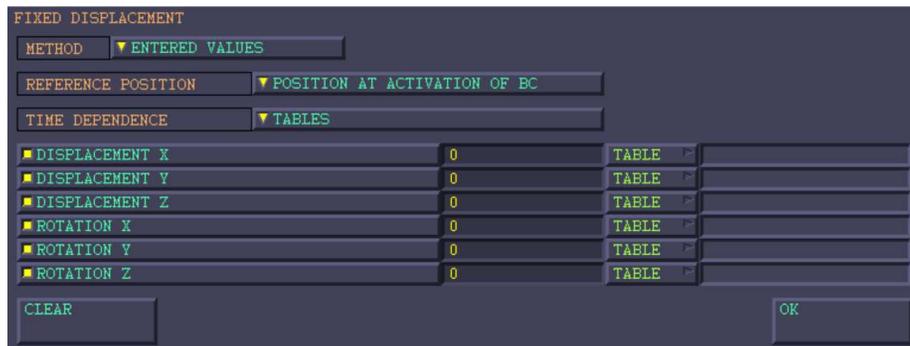
$u_0$  and  $F_0$  are the amplitude of the displacement or force

The resonance is defined as the condition when the  $\omega/\omega_n = 1$  in other words when the operating frequency corresponds to the natural frequency ( $\omega_n$ ) of the structure.

# BCs

Upper extremity fixed – lower extremity  $displ_z$  and  $rot_z$  free

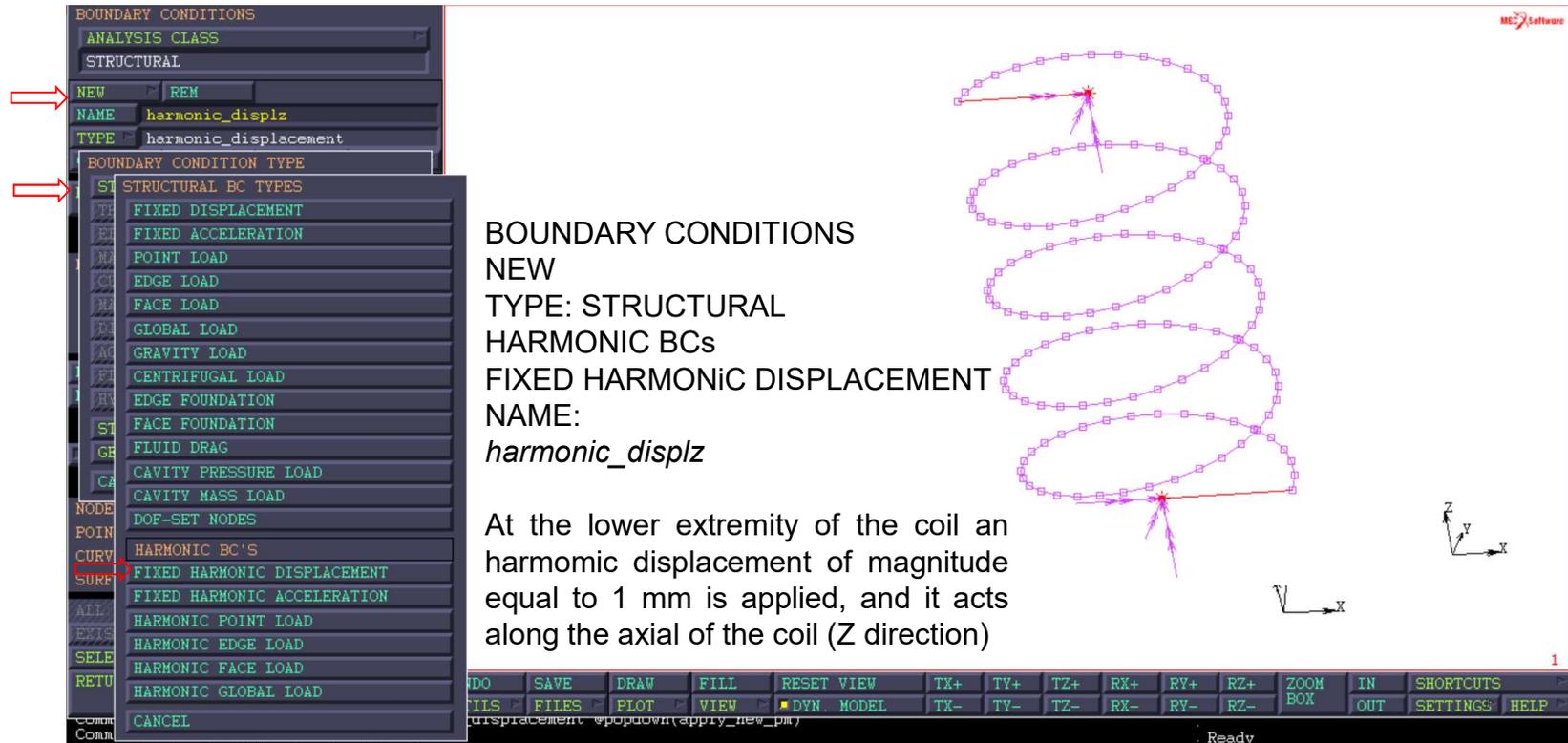
The natural frequencies of the coil are evaluated, considering the upper extremity of the coil as fixed, and the lower one as free to move and to rotate along Z direction. Therefore, the BCs are three and named *fixed*, *displrotz\_free* and *harmonic\_displz*.



# BCs

## Upper extremity fixed – lower extremity harmonic displ<sub>z</sub>

The natural frequencies of the coil are evaluated, considering the upper extremity of the coil as fixed, and the lower one as free to move and to rotate along Z direction. Therefore, the BCs are three and named *fixed*, *displrotz\_free* and *harmonic\_displz*.



BOUNDARY CONDITIONS

ANALYSIS CLASS  
STRUCTURAL

NEW REM

NAME harmonic\_displz

TYPE harmonic\_displacement

BOUNDARY CONDITION TYPE

STRUCTURAL BC TYPES

- FIXED DISPLACEMENT
- FIXED ACCELERATION
- POINT LOAD
- EDGE LOAD
- FACE LOAD
- GLOBAL LOAD
- GRAVITY LOAD
- CENTRIFUGAL LOAD
- EDGE FOUNDATION
- FACE FOUNDATION
- FLUID DRAG
- CAVITY PRESSURE LOAD
- CAVITY MASS LOAD
- DOF-SET NODES
- HARMONIC BC'S
- FIXED HARMONIC DISPLACEMENT
- FIXED HARMONIC ACCELERATION
- HARMONIC POINT LOAD
- HARMONIC EDGE LOAD
- HARMONIC FACE LOAD
- HARMONIC GLOBAL LOAD
- CANCEL

BOUNDARY CONDITIONS  
NEW  
TYPE: STRUCTURAL  
HARMONIC BCs  
FIXED HARMONIC DISPLACEMENT  
NAME:  
*harmonic\_displz*

At the lower extremity of the coil an harmonic displacement of magnitude equal to 1 mm is applied, and it acts along the axial of the coil (Z direction)

# BCs

Upper extremity fixed – lower extremity harmonic displ<sub>z</sub>

The natural frequencies of the coil are evaluated, considering the upper extremity of the coil as fixed, and the lower one as free to move and to rotate along Z direction. Therefore, the BCS are three and named *fixed*, *displrotz\_free* and *harmonic\_displz*.

BOUNDARY CONDITIONS  
ANALYSIS CLASS  
STRUCTURAL  
NEW REM  
NAME harmonic\_displz  
TYPE harmonic\_displacement  
COPY PREV NEXT EDIT  
PROPERTIES  
PLOT SETTINGS  
DRAW BOUNDARY CONDS ON MESH  
ID BOUNDARY CONDS  
ARROW PLOT SETTINGS  
MERGE DUPLICATE BOUND CONDS  
REMOVE ALL BOUND CONDS  
TABLES TRANSFORMATIONS  
NODES ADD REM 0  
POINTS ADD REM 0  
CURVES ADD REM 0  
SURFACES ADD REM 0  
ALL SELEC PISIB OUTL TOP  
EXIST UNSEL INVIS SURF BOT  
SELECT SET END LIST (\*)  
RETURN MAIN  
UNDO SAVE DRAW  
UTILS FILES PLOT

FIXED HARMONIC DISPLACEMENT  
METHOD ENTERED VALUES INPUT MODE MAGNITUDE & PHASE  
DISPLACEMENT X  
DISPLACEMENT Y  
DISPLACEMENT Z MAGNITUDE 1 TABLE  
PHASE (DEG) 0 TABLE  
ROTATION X  
ROTATION Y  
ROTATION Z  
CLEAR  
OK

BCS  
NEW  
TYPE: STRUCTURAL  
HARMONIC BCs  
FIXED HARMONIC DISPLACEMENT  
NAME:  
harmonic\_displz  
PROPERTIES  
 DIASPLACEMENT Z  
MAGNITUDE 1  
PHASE 0  
OK  
NODES: ADD  
Select the retained node of the  
rbe2\_z0.

Z+ ZOOM IN SHORTCUTS  
Z- BOX OUT SETTINGS HELP

# Loadcase

## Frequency response

The evaluation of the response of the structure is evaluated with a LOADCASE of DYNAMIC HARMONIC type.



# Loadcase

## Frequency response

Remember to update the INITIAL LOAD conditions at the LOADCASE menu, then submit the model to the evaluation.

LOADCASES

ANALYSIS CLASS  
STRUCTURAL

NEW REM

NAME *lcase5\_freq*

TYPE STRUCTURAL  
dyn\_harmo

COPY PREV NEXT EDIT

PROPERTIES

STRUCTURAL DYNAMIC HARMONIC LOADCASE PROPERTIES

LOADS

SOLUTION CONTROL

LOWEST FREQUENCY 10

HIGHEST FREQUENCY 1000

# FREQUENCIES 991

LOGARITHMIC INTERVALS

LOADCASE RESULTS

RESET OK

LOADCASES  
NEW  
DYNAMIC HARMONIC  
NAME:  
*Type lcase5\_freq*  
PROPERTIES  
LOWEST FREQUENCY 10  
HIGHEST FREQUENCY 1000  
# FREQUENCY 991  
 LOADS

DEACTIVATION / NC MACHINING

INPUT FILE TEXT INCLUDE FILE

TITLE TABLES

ALL SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

RY+ RZ+ ZOOM IN SHORTCUTS

RY- RZ- BOX OUT SETTINGS HELP

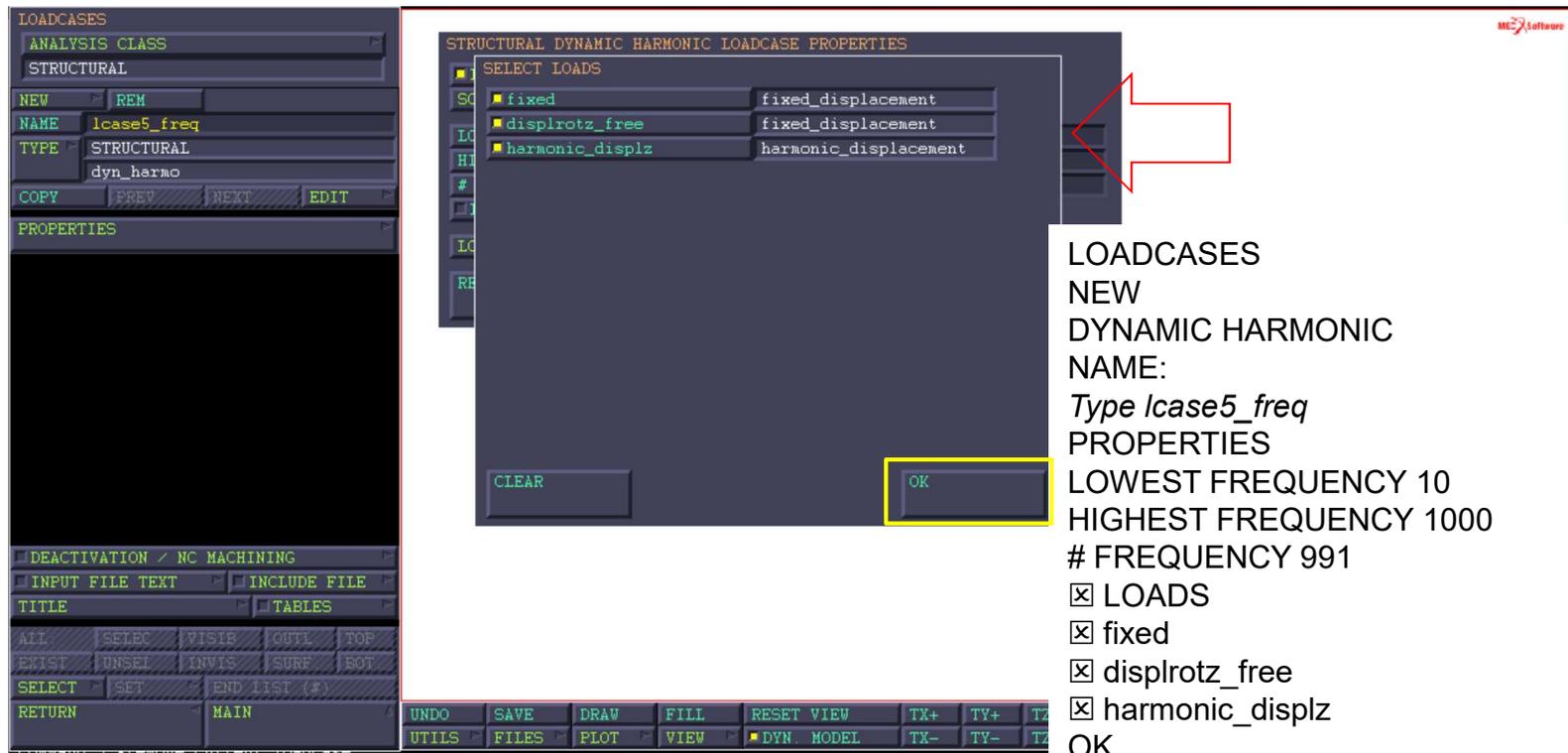
1

The frequency range that is investigated starts from 10 to 1000 Hz with a discrete step equal to 1 Hz.

# Loadcase

## Frequency response

Remember to update the INITIAL LOAD conditions at the LOADCASE menu.



LOADCASES

ANALYSIS CLASS  
STRUCTURAL

NEW REM  
NAME lcase5\_freq  
TYPE STRUCTURAL  
dyn\_harmo

COPY PREV NEXT EDIT

PROPERTIES

STRUCTURAL DYNAMIC HARMONIC LOADCASE PROPERTIES

SELECT LOADS

<input checked="" type="checkbox"/> fixed	fixed_displacement
<input checked="" type="checkbox"/> displrotz_free	fixed_displacement
<input checked="" type="checkbox"/> harmonic_displz	harmonic_displacement

CLEAR OK

LOADCASES  
NEW  
DYNAMIC HARMONIC  
NAME:  
*Type lcase5\_freq*  
PROPERTIES  
LOWEST FREQUENCY 10  
HIGHEST FREQUENCY 1000  
# FREQUENCY 991  
 LOADS  
 fixed  
 displrotz\_free  
 harmonic\_displz  
OK

And finally press OK on the previous shell.

# Jobs

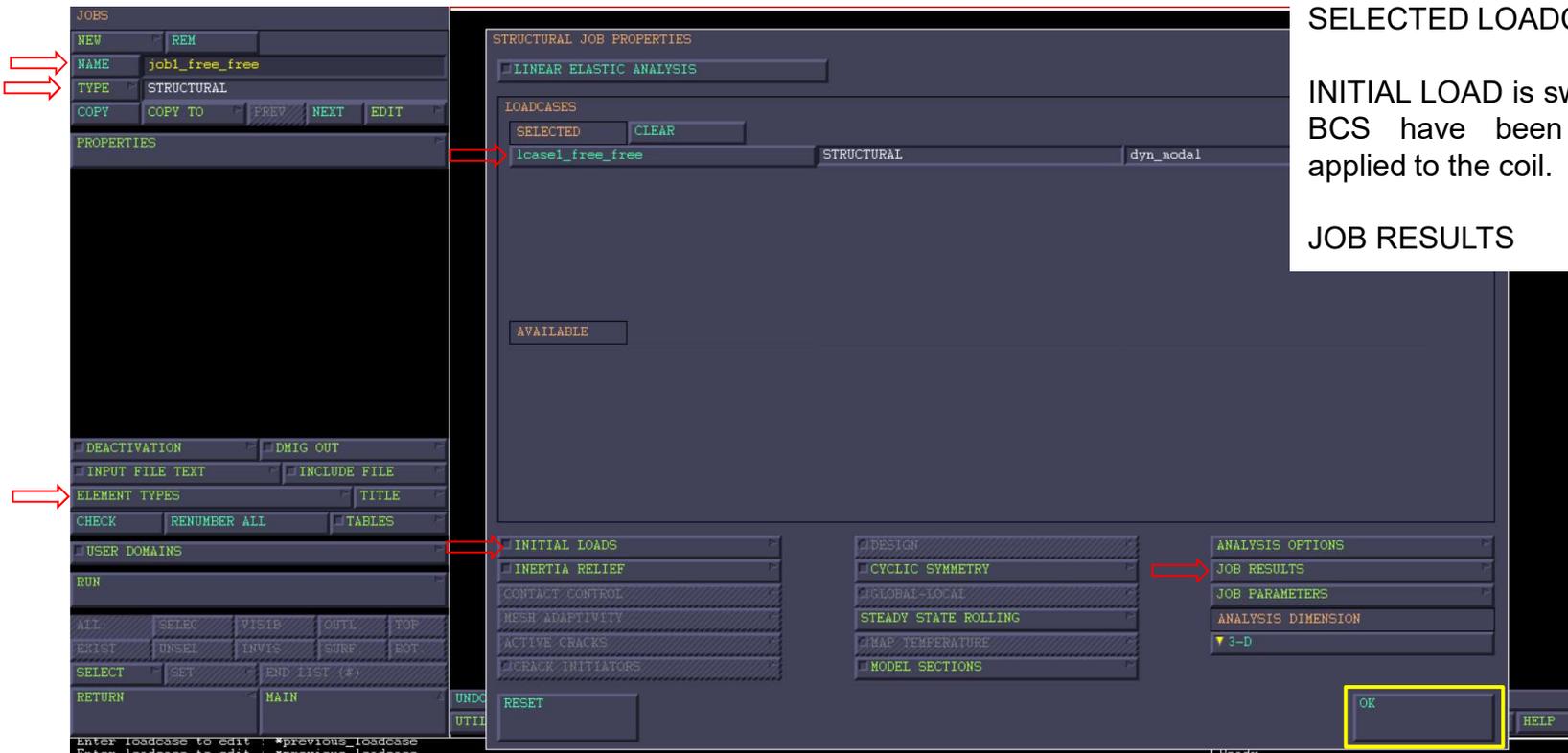
## Job results

JOBS  
NEW  
TYPE: STRUCTURAL  
NAME: job5\_freq  
PROPERTIES

The *lcase5\_freq* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

INITIAL LOAD is switched off, any BCS have been defined and applied to the coil.

JOB RESULTS



# Jobs

## Job results

JOB5  
 NEW  
 TYPE: STRUCTURAL  
 NAME: job5\_freq  
 PROPERTIES

The *lcase5\_freq* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

INITIAL LOAD is switched on, any BCS must be applied to the coil.

### JOB RESULTS

The equivalent real and the imaginary stress must be added

The screenshot displays the ANSYS Workbench software interface. On the left, the 'JOBS' panel shows a job named 'job1\_free\_free' with type 'STRUCTURAL'. Red arrows point to the 'NEW', 'NAME', and 'TYPE' fields. Below this, the 'PROPERTIES' panel is visible. In the center, the 'STRUCTURAL JOB PROPERTIES' dialog box is open, showing 'LINEAR ELASTIC ANALYSIS' and 'LOADCASES'. A red arrow points to the 'SELECTED' button, and another points to the 'lcase1\_free\_free' entry. Below the 'LOADCASES' section, the 'INITIAL LOADS' section is expanded, showing 'fixed' and 'displrotz\_free' entries. A red arrow points to the 'INITIAL LOADS' section. At the bottom of the dialog, the 'OK' button is highlighted with a yellow box. On the right, the 'ANALYSIS OPTIONS' panel is visible, with 'JOB RESULTS' selected. A red arrow points to the 'JOB RESULTS' option. At the bottom of this panel, the 'OK' button is also highlighted with a yellow box. The bottom of the screenshot shows the 'MILLE CHILI' logo and the date '23/05/2019'.

# Jobs

## Job results

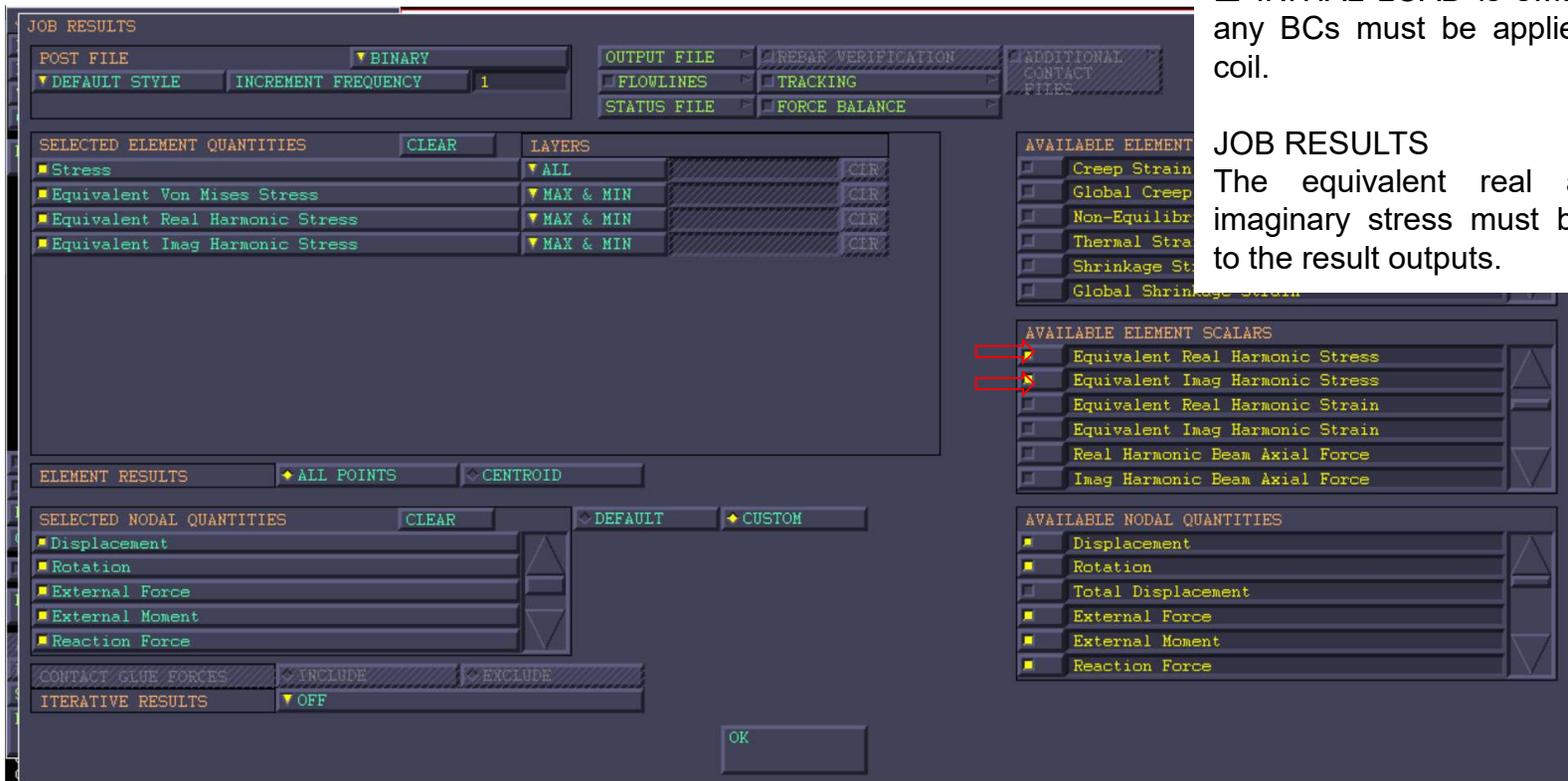
JOB  
NEW  
TYPE: STRUCTURAL  
NAME: job5\_freq  
PROPERTIES

The *lcase5\_freq* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

INITIAL LOAD is switched on, any BCs must be applied to the coil.

### JOB RESULTS

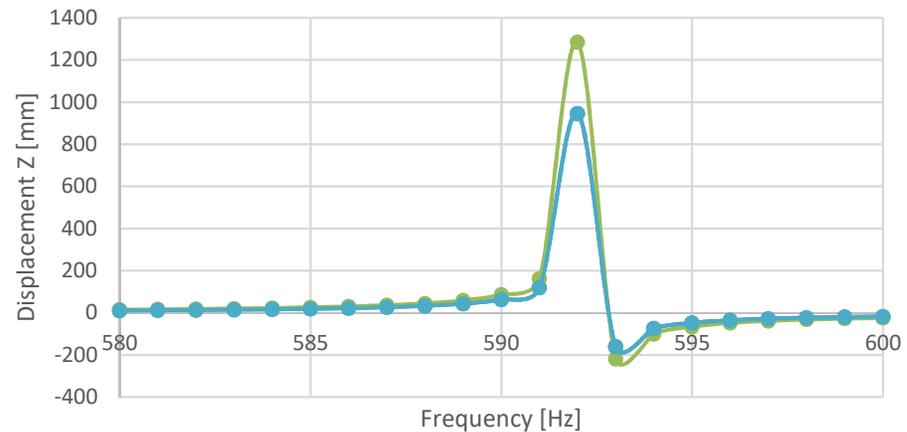
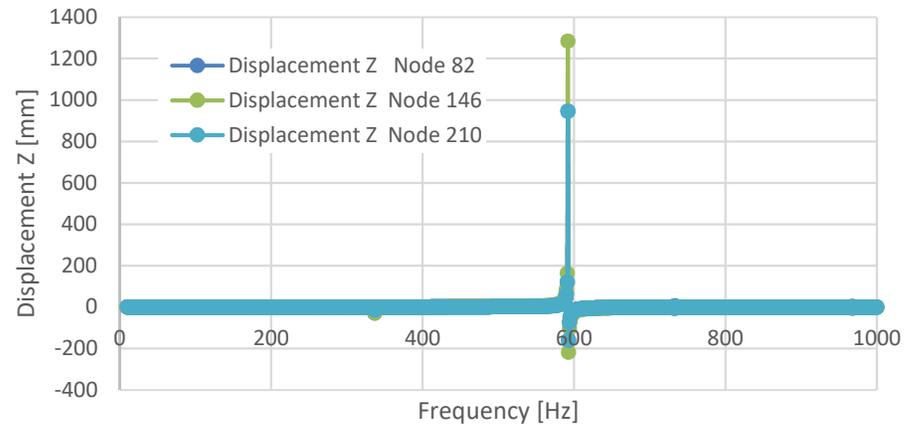
The equivalent real and the imaginary stress must be added to the result outputs.



# Results

## Frequency response: Hystory plot

The automatic collection of the frequencies of the coil (#991 frequencies) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files. The node of interest is the 82, 146 and 201 nodes, and the displacement Z is plotted on the frequency range under investigation [10-1000] Hz.



# Agenda

Goal and nomenclature

Mesh generation

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- **Considering damping effect**

References

# Material properties

## Isotropic and homogeneous with damping

The screenshot displays the ANSYS software interface for defining material properties. On the left, the 'MATERIAL PROPERTIES' dialog is open, showing 'STRUCTURAL' as the selected analysis class. The material name is 'titanium' and the type is 'standard'. The 'DATA CATEGORIES' section has 'STRUCTURAL' selected. The 'EXPERIMENTAL DATA FIT' section has 'SHOW MODEL' checked. The 'ELEMENTS' section shows 'ADD' and 'REM' buttons, with '144' elements listed. The 'REMOVE UNUSED MATERIALS' section is visible. The 'ALL' and 'EXIST' sections have 'SELEC' and 'UNSEL' buttons, respectively. The 'SELECT' section has 'SET' and 'END LIST (#)' buttons. The 'RETURN' section has 'MAIN' button.

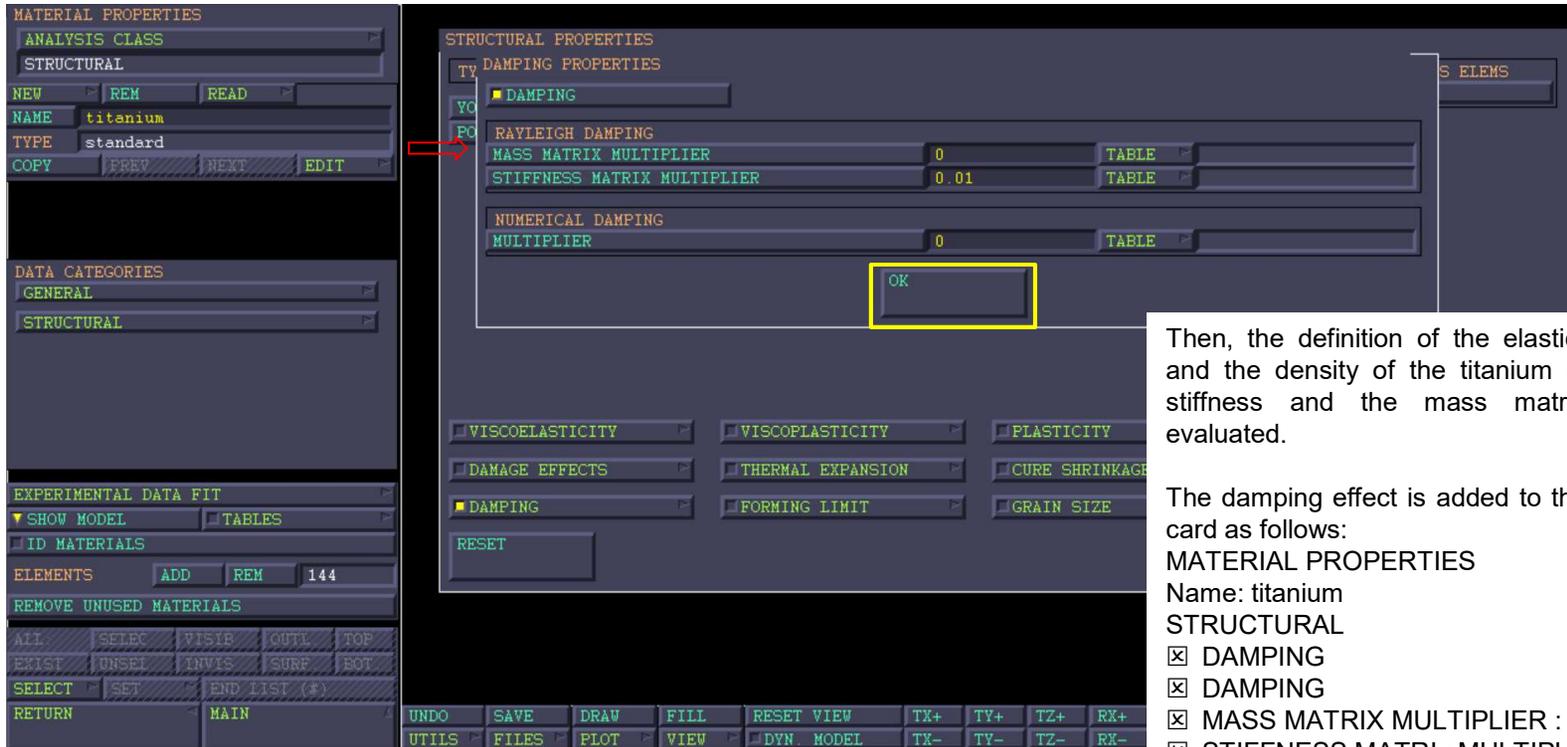
The right panel shows the 'STRUCTURAL PROPERTIES' dialog. The 'TYPE' is 'ELASTIC-PLASTIC ISOTROPIC'. The 'SHELL/PLANE STRESS ELEMS' section has 'UPDATE THICKNESS' checked. The 'YOUNG'S MODULUS' is 110000 and 'POISSON'S RATIO' is 0.3. The 'DAMPING' section is checked. The 'RESET' button is visible.

Then, the definition of the elastic constant and the density of the titanium allows the stiffness and the mass matrix to be evaluated.

The damping effect is added to the material card as follows:  
MATERIAL PROPERTIES  
Name: titanium  
STRUCTURAL  
 DAMPING

# Material properties

## Isotropic and homogeneous



Then, the definition of the elastic constant and the density of the titanium allows the stiffness and the mass matrix to be evaluated.

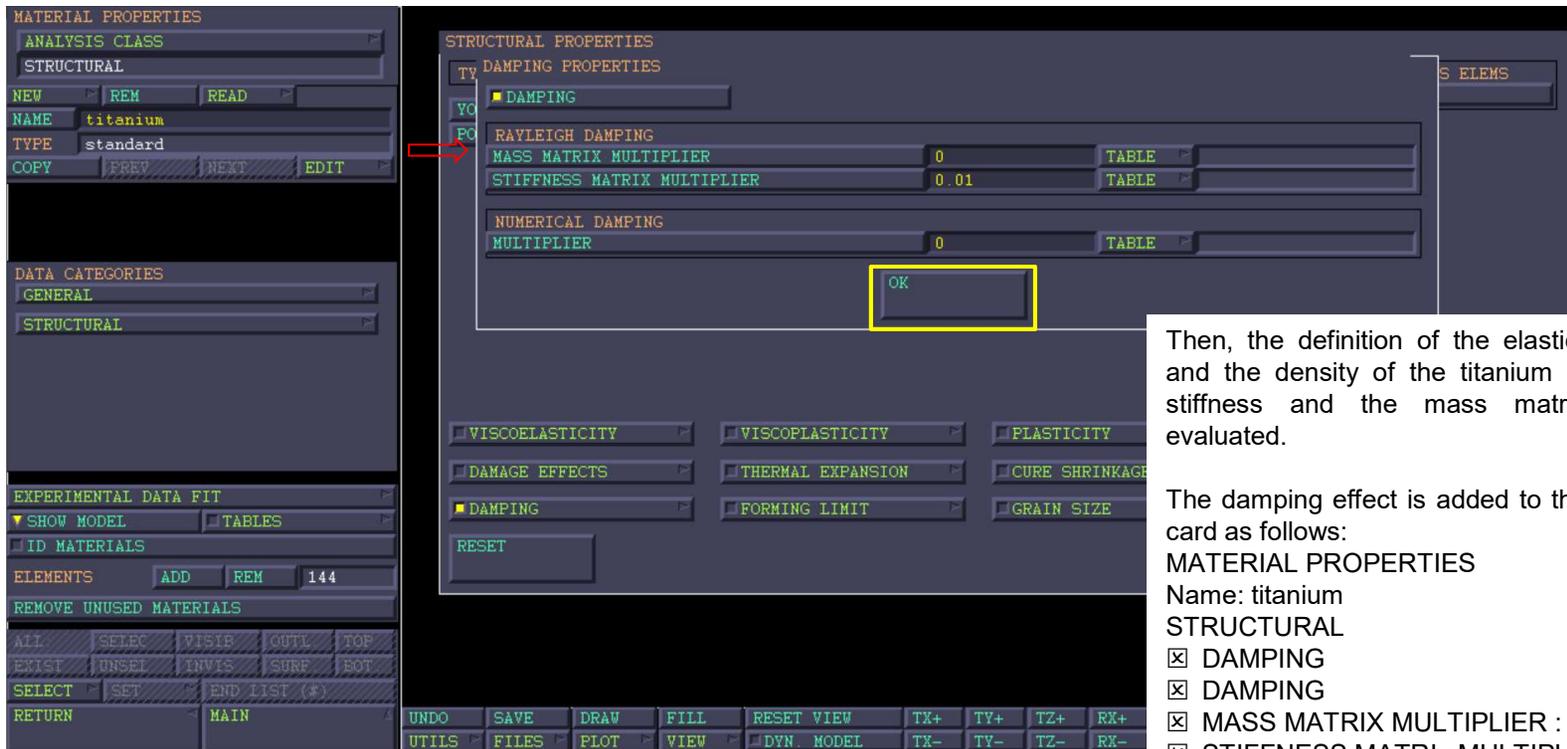
The damping effect is added to the material card as follows:

MATERIAL PROPERTIES  
 Name: titanium  
 STRUCTURAL  
 DAMPING  
 DAMPING  
 MASS MATRIX MULTIPLIER : 0  
 STIFFNESS MATRIX MULTIPLIER : 0.01  
 OK

The damping effect is consider fraction of the critically damper factor ( $\zeta$ ) adopting the Raileigh definition and assuming null the mass matrix multiplier ( $\alpha$ ) and equal to 1 per cent the stiffness matrix multiplier ( $\beta$ ).

# Material properties

## Isotropic and homogeneous with damping



Then, the definition of the elastic constant and the density of the titanium allows the stiffness and the mass matrix to be evaluated.

The damping effect is added to the material card as follows:

MATERIAL PROPERTIES

Name: titanium

STRUCTURAL

DAMPING

DAMPING

MASS MATRIX MULTIPLIER : 0

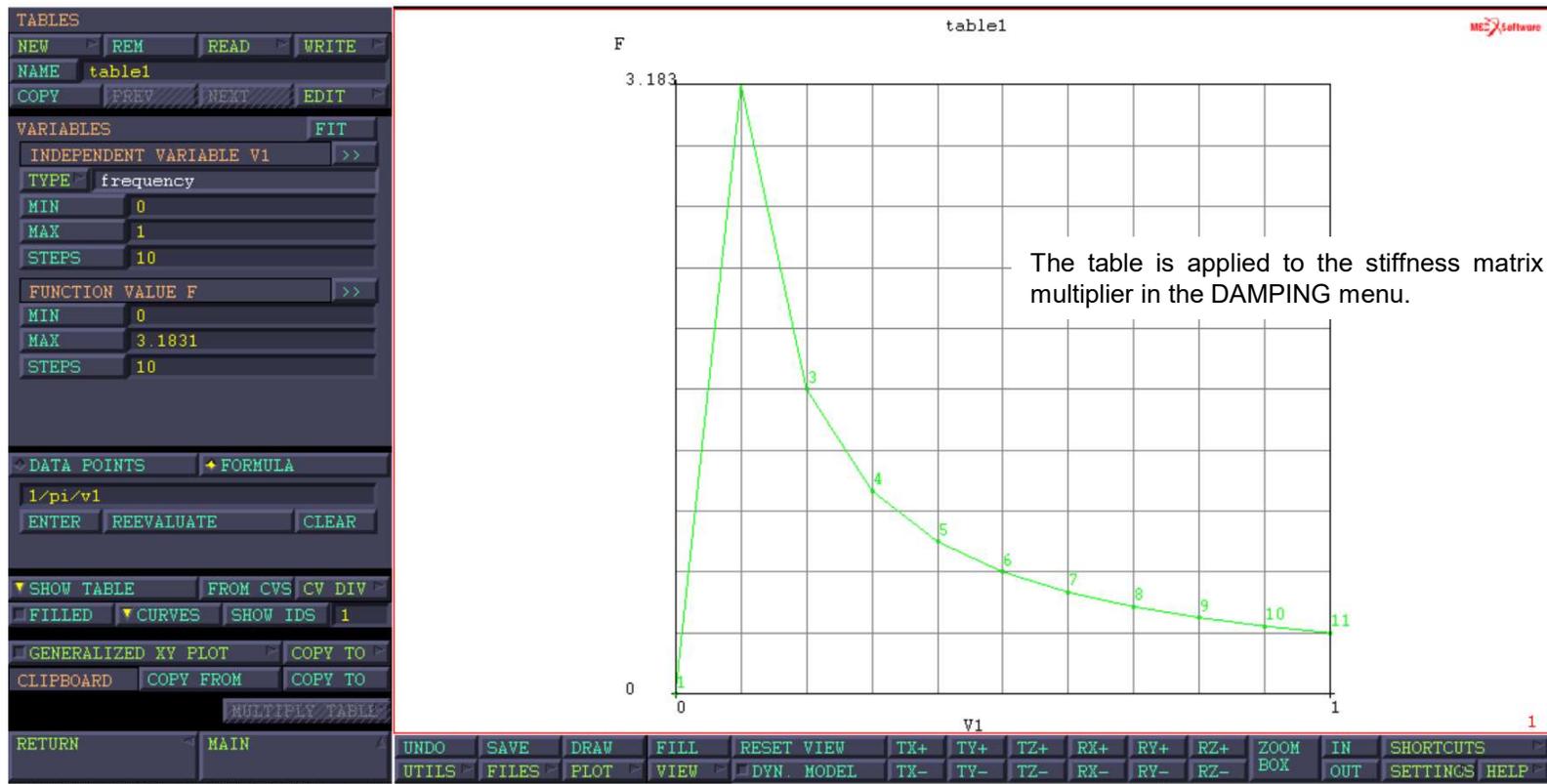
STIFFNESS MATRIX MULTIPLIER : 0.01

OK

The damping effect is considered a fraction of the critically damped factor ( $\zeta$ ) adopting the Rayleigh definition and assuming null the mass matrix multiplier ( $\alpha$ ) and equal to 1 per cent the stiffness matrix multiplier ( $\beta$ ).

# Material properties

Isotropic and homogeneous with damping



# Jobs

## Job results

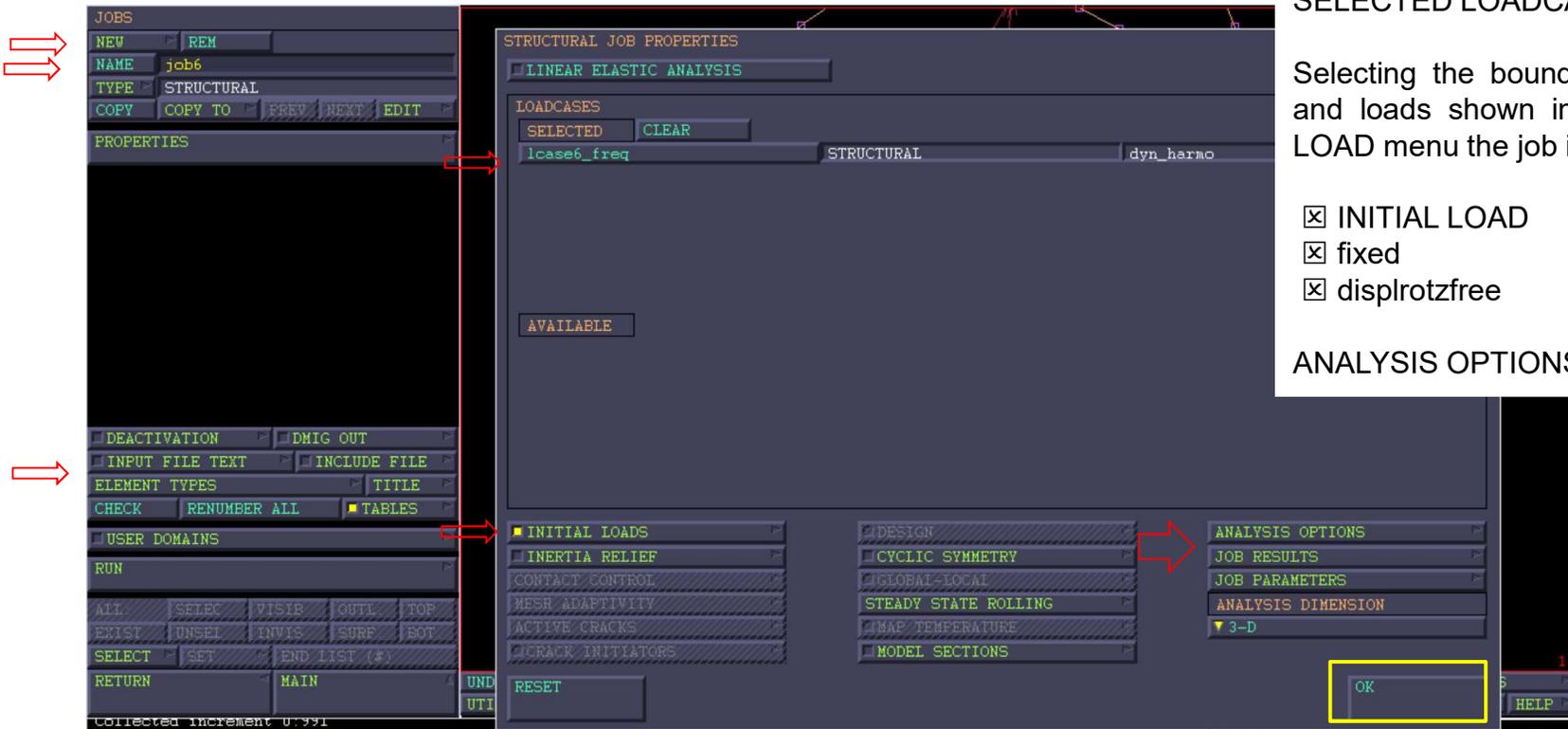
JOB  
NEW  
TYPE: STRUCTURAL  
NAME: job2\_fixed\_fixed  
PROPERTIES

The *lcase6\_freq* must be updated from the AVAILABLE to the SELECTED LOADCASES option.

Selecting the boundary condition and loads shown in the INITIAL LOAD menu the job is defined.

- INITIAL LOAD
- fixed
- displrotzfree

ANALYSIS OPTIONS



# Jobs

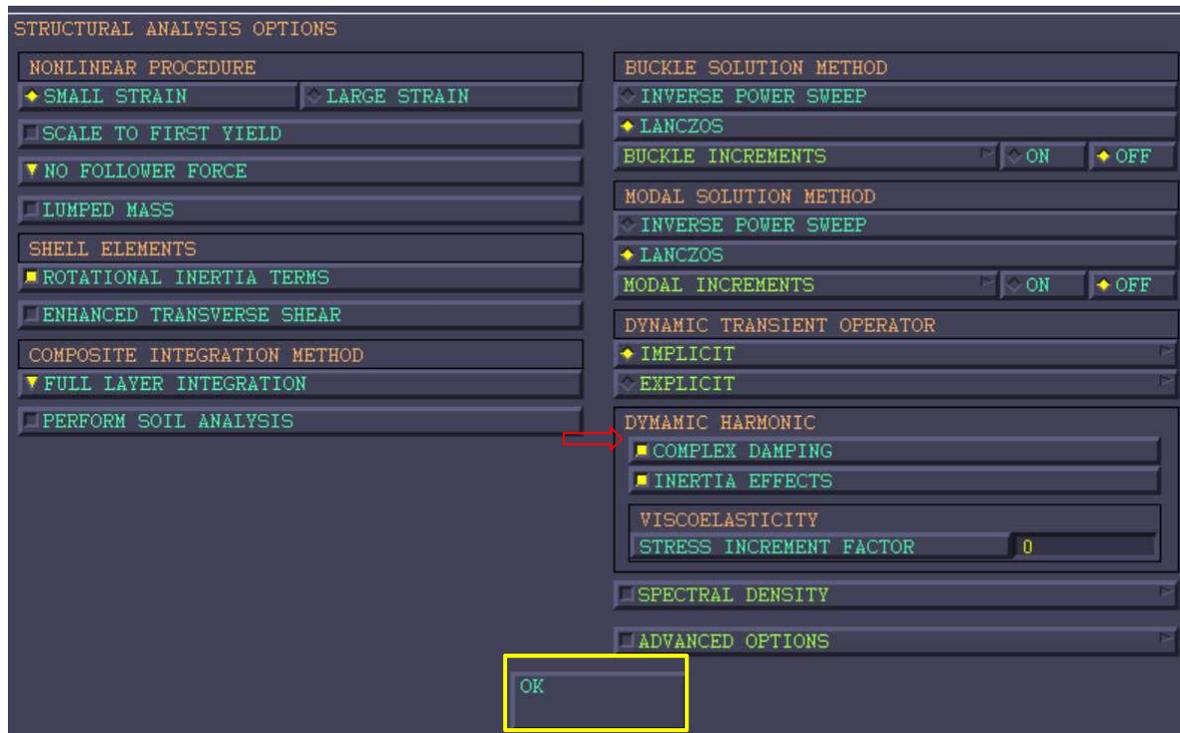
## Job results

ANALYSIS OPTION  
 COMPLEX DAMPING

OK

We enable the complex damping effects in the solution of the dynamic problem.

JOB RESULTS .



# Jobs

## Job results

ANALYSIS OPTION  
 COMPLEX DAMPING

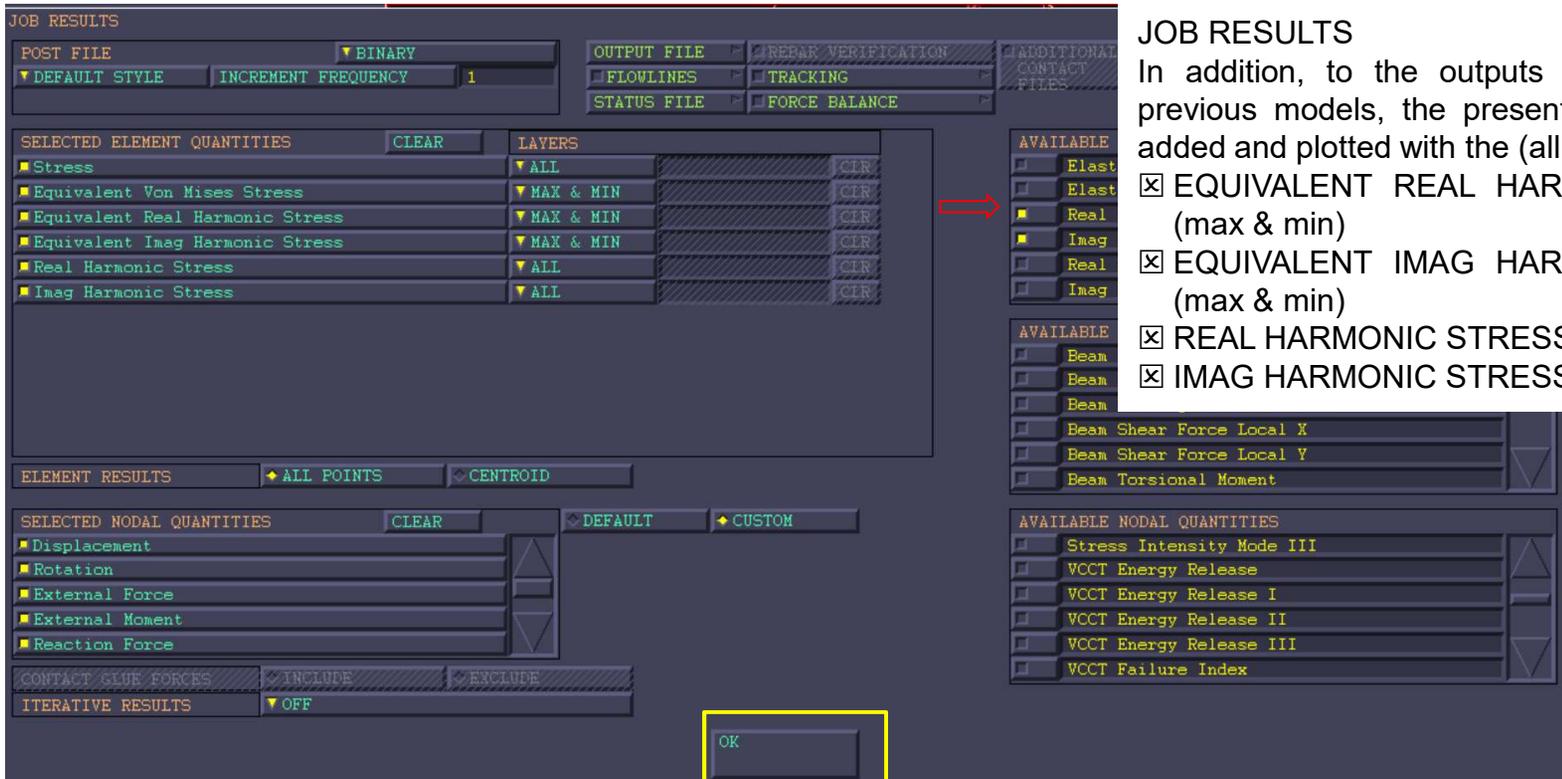
OK

We enable the complex damping effects in the solution of the dynamic problem.

### JOB RESULTS

In addition, to the outputs requested in the previous models, the present results must be added and plotted with the (all or max & min):

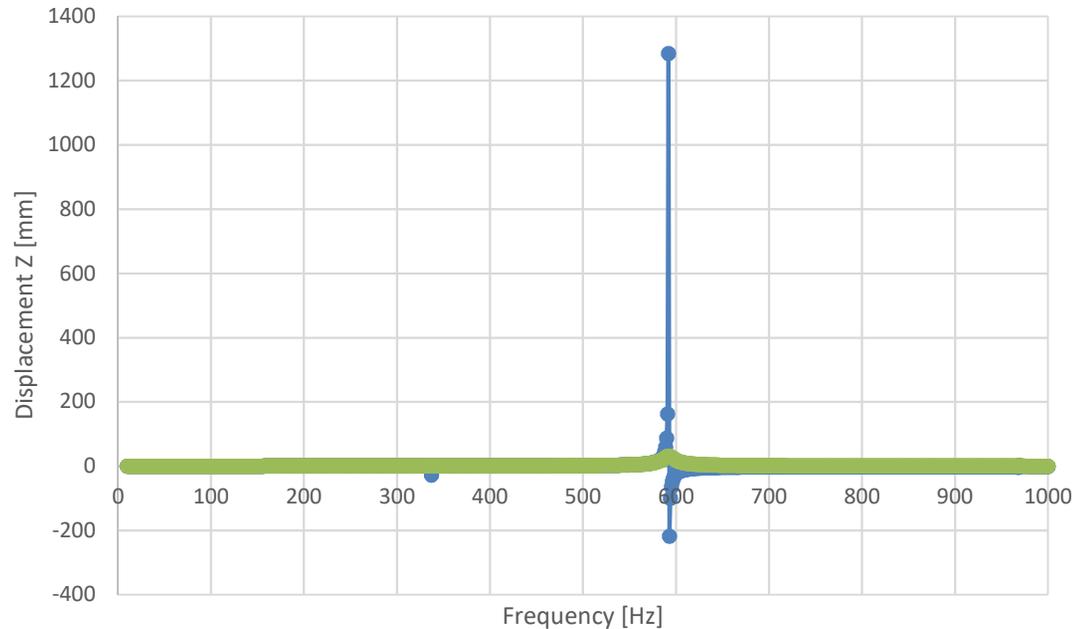
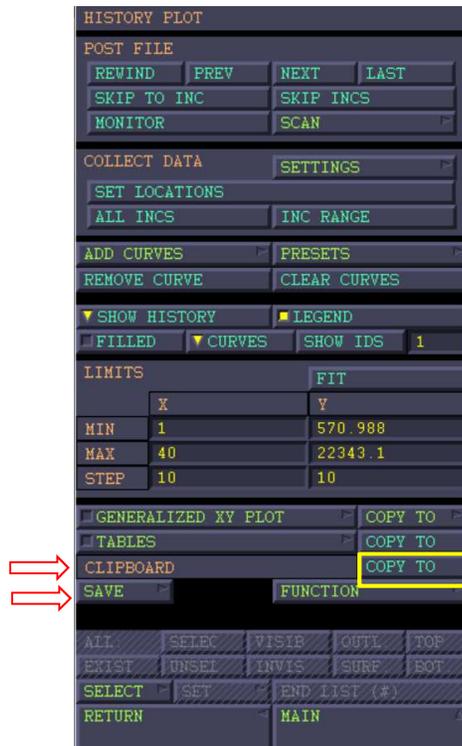
- EQUIVALENT REAL HARMONIC STRESS (max & min)
- EQUIVALENT IMAG HARMONIC STRESS (max & min)
- REAL HARMONIC STRESS (all)
- IMAG HARMONIC STRESS (all)



# Results

## Frequency response: Hystory plot

The automatic collection of the frequencies of the coil (#991 frequencies) can be done by using HYSTORY PLOT option. These data can usually exported to make the post-processing in a Spreadsheet file, e.g. .xls, .xlsx, .ods, ... files. The node of interest is the 146 node, and the displacement Z is plotted on the frequency range under investigation [10-1000] Hz.



● Displacement Z Node 146  
Without damping

● Displacement Z Magnitude Node 146  
With damping

# Results

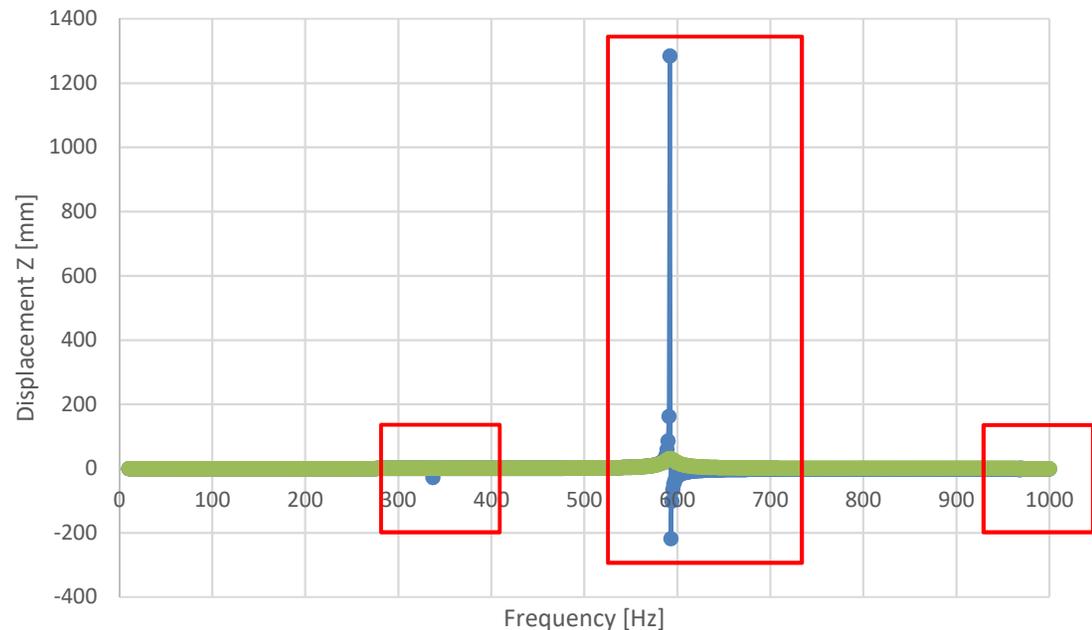
## Frequency response: Hystory plot

At the natural frequencies collected by the modal analysis (*3\_modal\_fixed\_rotzfree.mud*), the frequency response analysis evidences a fluctuation of the Z-displacement in sign.

Comparing the two models that consider or neglect the damping, the frequency response amplitude becomes finite, e.g. at 592 Hz the peak is 31.5 Hz (see Excel file).

Considering damping the #3 and #4 natural modes collected in the model *3\_modal\_fixed\_rotzfree.mud* are covered by the #2 mode. The damping affects the frequency response by mitigating some peaks.

3_modal_fixed_rotzfree.mud		
Sub Increment	Frequency	
1	337.0	
2	592.1	
3	726.4	
4	732.6	
5	968.5	



● Displacement Z Node 146  
Without damping

● Displacement Z Magnitude Node 146  
With damping

# Agenda

Goal and nomenclature

Mesh generation

Modal Analysis loadcases varying the BCs:

- Free-free
- Fixed-fixed
- Fixed-axial rotation free
- Fixed-axial displacement and rotation free

Harmonic loadcases:

- Neglecting damping effect
- Considering damping effect

**References**

# References

## **Books:**

Garro A. Progettazione strutturale del motore, Levrotto & Bella, Torino, 1992. pp. 464-465

Gugliotta A. Elementi Finiti, progetto didattica in rete, Otto Editore, 2002. parte IV

## **FE models and procedures:**

mesh\_geom\_mat\_link.mud (.proc)

→ Using the present .proc the starting model is defined and recalled for setting the further analyses.

### *For modal analysis:*

1\_modal\_free\_free.mud (.proc)

2\_modal\_fixed\_fixed.mud (.proc)

3\_modal\_fixed\_rotzfree.mud (.proc)

4\_modal\_fixed\_displrotzfree.mud (.proc)

### *For frequency response analysis:*

5\_freq\_fixed\_displrotzfree.mud (.proc)

6\_freq\_fixed\_displrotzfree\_damping.mud (.proc)

## **Excel file:**

modal\_frequency\_analysis\_coil.xlsx

→ post-processing result

# References

## **Video:**

### **Engine valve distribution**

<https://vehiclecue.it/fasatura-variabile-descrizione-e-funzionamento/9552/>

### **Bridge**

<https://www.youtube.com/watch?reload=9&v=3mclp9QmCGs>

### **Helicopter**

<https://www.youtube.com/watch?v=-LFLV47VAbI>