

**A**UTOMATIC

**D**YNAMIC

**I**NCREMENTAL

**N**ONLINEAR

**A**NALYSIS

ADINA System 8.7  
Release Notes

May 2011

ADINA R & D, Inc.

# **ADINA System 8.7**

## Release Notes

(including improvements up to 8.7.4)

May 2011

ADINA R & D, Inc.  
71 Elton Avenue  
Watertown, MA 02472 USA

tel. (617) 926-5199  
telefax (617) 926-0238  
[www.adina.com](http://www.adina.com)

---

## **Notices**

ADINA R & D, Inc. owns both this software program system and its documentation. Both the program system and the documentation are copyrighted with all rights reserved by ADINA R & D, Inc.

The information contained in this document is subject to change without notice.

ADINA R & D, Inc. makes no warranty whatsoever, expressed or implied that the Program and its documentation including any modifications or updates are free from errors or defects. In no event shall ADINA R&D, Inc. become liable to the User or any party for any loss, including but not limited to, loss of time, money or goodwill, which may arise from the use of the Program and its documentation including any modifications and updates.

## **Trademarks**

ADINA is a registered trademark of K.J. Bathe / ADINA R & D, Inc.

All other product names are trademarks or registered trademarks of their respective owners.

## **Copyright Notice**

© ADINA R & D, Inc. 2010-2011  
May 2011 Printing  
Printed in the USA

---

# New and updated feature summary

This section lists the new and updated features that are available in ADINA System 8.7.0, as compared with ADINA System 8.6. (For features and improvements added between 8.7.0 and 8.7.4, see the section "Improvements between 8.7.0 and 8.7.4" below.)

There are new commands and new and changed parameters associated with the new and updated features. The release notes refer to the commands and parameters in the command-line formats. Further information about the new commands and new and changed parameters can be found in the AUI Command Reference Manuals.

For user interface users, most command-line parameters have analogous fields in the dialog boxes.

Note, when we refer to documentation, we refer to the versions of the documentation given below in the "Available Documentation" section.

## Features for all programs

**Supported platforms:** The table on the next page gives a list of all supported program versions.

## ADINA Solids & Structures features

**Implicit DMP solver:** ADINA 8.7 can run in distributed memory processing (DMP) mode, on multiple networked computers. This feature is currently available for x86\_64 processors running Linux. Three solvers currently support DMP mode: the sparse solver, the 3D-iterative solver, and the Subspace eigensolver. ADINA, ADINA-CFD and ADINA-FSI analyses that use these solvers will benefit from DMP parallelization. DMP can also be combined with the shared memory processing (SMP) parallelization already existing in ADINA.

**Improvements to restart:** The restart file can now store a user-specified number of steps. There is now an analysis mode in which the last solution in a restart file can be retrieved, the results from this last solution printed or saved, then the program stops.

➤ Command-line:

```
MASTER ... IRSTEPS  
MASTER ... MODEX=RESULTS
```

**Improvements to ADINA TMC:** The following improvements are made to ADINA TMC:

- Piezoelectric and consolidation can now be used in the ADINA-TMC solution option, which also means that these features can now be used in ADINA-FSI problems. Piezoelectric material models are provided as user-supplied models and can be modified by users.

**Table of supported program versions**

Platform	Operating system	Version	ADINA-M	Parallelized assembly <sup>1</sup>	Fortran compiler
Linux Itanium	2.6.9 and higher, glibc 2.3.4 and higher	64-bit	Yes <sup>2</sup>	Yes	Intel ifort 10.1
Linux x86	2.4.20 and higher, glibc 2.3.2 and higher <sup>3</sup>	32-bit	Yes	Yes	Intel ifort 8.1
Linux x86_64 / Opteron	2.6.9 and higher, glibc 2.3.4 and higher	64-bit	Yes	Yes	Intel ifort 8.1
IBM AIX	AIX 5.1	64-bit	Yes	No	IBM xlf90 8.1
PC x86	Windows XP, Vista, 7	32-bit <sup>4</sup>	Yes	No	Compaq Visual Fortran 6.6
PC x86_64 / Opteron	Windows XP, Vista, 7	64-bit	Yes	Yes	Intel Visual Fortran 11

1) All program versions have parallelized solvers. Only ADINA and ADINA-Thermal have parallelized assembly.

2) The x86 version of the AUI is provided so that ADINA-M can be used.

3) Red Hat 9.0 or higher, or equivalent.

4) The 3GB address space option is supported.

**Improvements to glueing:** Birth and death times can now be specified for glued surfaces and lines.

- Command-line:  
GLUEMESH ... TBIRTH TDEATH

**Prescribed velocity and acceleration loads:** Prescribed velocities and prescribed accelerations can now be applied in linear and nonlinear dynamic analysis, when the following time integration methods are used:

- implicit time integration using the Newmark and Bathe composite methods
- explicit time integration

- Command-line:  
LOAD VELOCITY (new command for ADINA)  
LOAD ACCELERATION (new command)  
APPLY-LOAD

**Solution diagnostics:** The program can output, after each equilibrium iteration in static or implicit dynamic analysis, useful information about the solution. This information includes nodes coming into or out of contact, element integration points switching to and from plasticity, etc. This information can help in understanding the reason if the solution does not converge.

- Command-line:  
DIAGNOSTICS SOLUTION=DETAILED (new command)

**Warping-capable beam element:** A new beam element is introduced that takes into account the warping displacement of the cross section of the beam and its effect on the axial strains. This element can be used to model general thin-walled elastic open section beams e.g. I-beam, U-beam (channel section) and L-beam (angle section) in general linear and large deformation nonlinear analysis. The element is particularly useful in modeling instability phenomena such as lateral buckling and axial-torsional buckling. Nonlinear kinematics due to torsion (the so-called Wagner effect) is also included in the formulation.

- Command-line:  
EGROUP BEAM ... WARP=YES  
FIXITY  
  dof<sub>i</sub> = BEAM-WARPING  
BOUNDARIES  
  ... beam-warping<sub>i</sub>  
CROSS-SECTION PROPERTIES ... CTOFFSET CSOFFSET STINERTIA  
  SRINERTIA TRINERTIA WINERTIA  
  WRINERTIA DRINERTIA

**3D-shell element:** A new 3D-shell element is introduced that is suitable for the solution of large strain out-of-plane bending problems, for example, crushing of thin-walled tubes. The element can be used in static, frequency, implicit dynamic and explicit dynamic analyses. The elastic-isotropic, plastic-orthotropic, plastic-cyclic and rubber material models are supported, under small or large strain conditions.

- Command-line:  
EGROUP SHELL ... SHELLTYPE=3DSHELL

**Improvements to thermo-plastic and thermo-plastic-creep material models:** Orthotropic thermal coefficients can now be specified for thermo-plastic and thermo-plastic-creep material models for small and large displacements (but small strains). This option is available for 2-D, 3-D and shell elements.

- Command-line:  
MATERIAL MULTILINEAR-PLASTIC-CREEP ... THERMAL-EXPANSION

**Improvements to viscoelastic material models:** Orthotropic thermal coefficients can now be specified for the viscoelastic material model for small and large displacements (but small strains). This option is available for 2-D, 3-D and shell elements.

- Command-line:  
MATERIAL VISCOELASTIC ... THERMAL-EXPANSION

**Improvement to user-coded material models:** For the Linux versions, the user-coded object to use for a run can be chosen at run-time, using the new `-usrlib` option of the `adina8.7` script.

**op2 file improvements:** Op2 files can be requested for ADINA models set up using the AUI, for the following element and analysis types:

Truss, 2D solid, 3D solid, beam, shell and spring elements

Static, dynamic and frequency results, except for modal participation factors calculation

The substructuring and cyclic symmetry analysis features are not supported.

**Femap neutral files:** Femap neutral files can be requested for ADINA models.

- Command-line:  
MASTER ... RESULT=NEU

## ADINA CFD & FSI features

**Implicit DMP solver:** ADINA-F 8.7 can run in distributed memory processing (DMP) mode. This feature is very similar to the implicit DMP solver feature of ADINA 8.7, see above for more details.

**Periodic boundary conditions:** Periodic boundary conditions can be used when the physical geometry and the expecting flow pattern have a periodicity in space. Each boundary condition is specified as a pair in which one boundary is transformed from the other.

- Command-line:  
BOUNDARY-CONDITION PERIODIC (new command)

**One-way thermal coupling through FSI boundary conditions:** In this thermal coupling technique, the temperature in the fluid model at the FSI boundary condition is passed to the solid model. It is assumed that the structural solution does not affect the fluid solution.

- Command-line:  
MASTER ... THERMAL-FSI=TQ

## AUI features

### User interface

**Improvements to Model Tree View:** General constraints, glue mesh and initial conditions on nodes are added to the model tree. Zones are also listed in the model tree and can be displayed or highlighted easily using the right mouse button.

**Improvements in entity picking:** For many dialog boxes, picking of entities is greatly improved by highlighting all picked entities (table entry picking) instead of only the last picked entity. Marquee picking mode is also allowed and the “S” key (subtract) can be used to unpick entities.

**Convergence graph:** For nonlinear ADINA Structural analysis, convergence graphs for all steps are saved (in .ite file) and available for viewing during and after analysis.

**EXECUTE command:** The EXECUTE command can be used to invoke a shell command from within the AUI (UNIX/Linux versions only).

- Command-line:  
EXECUTE (new command)

## Model definition

**Redefinition of local coordinate systems:** The default for the SYSTEM ... MOVE parameter is changed from NO to YES.

- Command-line:  
SYSTEM ... MOVE

**Mesh transformation:** When transforming geometry, the mesh associated with the geometry can also automatically be transformed.

- Command-line:  
LINE TRANSFORMED ... MESH EGROUP  
SURFACE TRANSFORMED ... MESH EGROUP  
VOLUME TRANSFORMED ... MESH EGROUP  
BODY TRANSFORMED ... MESH EGROUP

**Improvements in Nastran file export:** ADINA Thermal models can be exported.

**Removal of small features from bodies:** A Parasolid body can be defeatured, that is, small features (faces) can be removed from the body.

- Command-line:  
BODY DEFATURE (new command)

**Saving of the Parasolid file within the AUI database file:** The Parasolid file is now automatically saved within the AUI database file (extension .idb).

**Additional methods for creating node-sets:** Node-sets can be created based on the search distance from a specified node.

- Command-line:  
NODESET ... OPTION=NEARBY DISTANCE

**Additional uses for node-sets:** Spring elements can be created using node-sets.

- Command-line:  
SPRING NODESETS (new command)

**Generation of hexahedral meshes with GHEXA:** Non-uniform mesh densities can be assigned to body edges and faces as in general body meshing (with GBODY). GHEXA also supports curvature-based refinement (GEO-ERROR, SAMPLING, MIN-SIZE), automatic grading (AUTO-GRADING) and simulated meshing (SIMULATE). The quality of

quadrilateral facets can be controlled by QFCALIM and QFCDALIM. By setting SIMPLIFY to YES, the relationship between the mesh and the body's topology is loosened and better quality meshes can be obtained.

➤ Command-line:

```
GHEXA ... QFCALIM QFCDALIM GEO-ERROR SAMPLING MIN-SIZE  
AUTO-GRADING SIMULATE SIMPLIFY
```

**Cleanup of Parasolid parts:** Edges (PHASE=SPLITE) and faces (PHASE=SPLITF) can be split in order to allow the removal of thin faces (PHASE=REMOVE).

➤ Command-line:

```
BODY-CLEANUP ... PHASE
```

**Generation of quadrilateral meshes with GFACE:** Pre-processing of the subdivisions on the bounding edges of a body face. If EVEN is set to ALL, all bounding edges are forced to have an even number of subdivisions.

➤ Command-line:

```
GFACE ... EVEN
```

## **Model display and post-processing**

**Improvement for mass properties listings:** In 8.6 and earlier, when listing mass variables using ZONELIST, the mass variables were based on the zone used in the MASS-SELECT command, not the zone used in the ZONELIST command. In 8.7, the mass variables listed using ZONELIST are based on the zone used in the ZONELIST command. (The same applies for ZONEMAX and ZONEEXCEED as well.)

**Improvement for graph legends:** The default graph legends are changed when creating a time history graph using RESPONSESHOW. In 8.6 and earlier, the default graph legend was the text 'NO LEGEND'. In 8.7, the default graph legend includes the y variable and y point.

**Improvement for cutting surfaces:** Cutting surfaces now process 3- and 4-node shell elements that are drawn in the topbottom depiction.

**Improvement for plotting of material properties for rubber-like materials:** The default number of points used to plot the stress-strain curves for rubber-like materials is increased, when there are many points in the curves used for curve-fitting.

**RESPALL option for ZONEMAX and LINEMAX:** In 8.6 and earlier, the ZONEMAX and LINEMAX commands listed the maximum value of each variable, considering all responses in the response-range. Using the new RESPALL option, these commands list the maximum value of each variable, for each response in the response-range.

➤ Command-line:

```
ZONEMAX ... RESPOPTION=RESPALL  
LINEMAX ... RESPOPTION=RESPALL
```

**Support for beam warping:**

- New variables are added: BEAM\_WARPING, BEAM\_WARPING\_VELOCITY, BEAM\_WARPING\_ACCELERATION, BEAM\_WARPING\_EIGENVECTOR, BIMOMENT, BIMOMENT\_REACTION, NODAL\_BIMOMENT. These variables can be listed or plotted as bands.
- The warping fixity is added to fixity plotting.
- Element lines of warping displacement, bimoment, etc can be plotted as element line plots.

**Support for prescribed velocities and accelerations:**

- New variables are added: {XYZ}-PRESCRIBED\_ANG\_VELOCITY, {ABC}-PRESCRIBED\_ANG\_VELOCITY, {V1, V2}-PRESCRIBED\_ANG\_VELOCITY
- Prescribed velocity and acceleration loads can be plotted.

## **TRANSOR features**

**TRANSOR for I-deas:** NX I-deas 5 and 6 are supported on Windows.

# Improvements between 8.7.0 and 8.7.4

## ADINA Solids & Structures improvements

### Stiffness stabilization and mesh glueing

In 8.7.0, stiffness stabilization is used whenever there is mesh glueing, as follows:

MASTER STABILIZATION=NO:

stabilization factor 1E-10 is used

STABILIZATION=YES:

user-specified stabilization is used

STABILIZATION=AUTOMATIC:

stabilization is used when necessary, the larger of the user-specified stabilization and 1E-10 is used.

The default is MASTER STABILIZATION=NO. Therefore stabilization is used by default in 8.7.0.

For certain meshes, this default stabilization factor changes the stiffness matrix significantly. For linear analysis, the solution is changed and for nonlinear analysis, the convergence rate is made worse.

To avoid this problem, in 8.7.1 and higher, the logic is changed as follows:

MASTER STABILIZATION=NO:

no stabilization factor

STABILIZATION=YES:

user-specified stabilization is used

STABILIZATION=AUTOMATIC:

stabilization is used when necessary, the user-specified stabilization is used.

This logic is the same as if mesh glueing is not used.

So in 8.7.1 and higher, when mesh glueing is used, the model might be unstable if the glued parts are individually unstable when glueing is not considered. In that case, stiffness stabilization should be turned on, using the smallest stabilization factor necessary to obtain a solution.

### **Sparse solver**

The sparse solver for the Linux 64-bit version now runs 10-20% faster in 8.7.2 and higher.

### **Modal damping ratio calculations**

See the section "Modal damping ratio calculations" below for more information.

### **Direct matrix input**

Stiffness matrices (and/or mass and damping matrices for dynamic analysis) can be directly provided in 8.7.3 and higher. The matrices are node-based and can be used to connect any nodal points in the model.

See new command DMIG-DEFINE description for details.

### **Export of matrices in terms of nodes**

The stiffness matrix (also mass and damping matrices in dynamic analysis) can now be exported in terms of nodes in 8.7.3 and higher. Previously these matrices could be exported in terms of equations only. The option can be accessed via the existing DISK-STORAGE command:

```
DISK-STORAGE GLOBAL-MATRIX=NODE STIFFNESS=SAVE ,  
                                     MASS-MATRIX=SAVE ,  
                                     DAMPING-MATRIX=SAVE
```

### **Concentrated masses and constraint equations**

The information below applies to constraint equations in which the slave degree of freedom is constrained to more than one master degree of freedom. The following improvements are made for 8.7.1 and higher.

If a consistent mass matrix is used and concentrated masses are applied to constrained nodes, then the off-diagonal terms are included in the global mass matrix.

Similarly, if concentrated dampers are applied to constrained nodes, then the off-diagonal terms are included in the global damping matrix.

### **Initial temperatures in restart analysis**

In 8.7.1 and higher, it is allowed to update the initial temperatures in a restart analysis (command MASTER ... INITIALTEMP=YES). This option is not the default.

### **Internal heat generation**

In 8.7.1 and higher, internal heat generation can be temperature and time dependent.

### **Writing of sparse matrices**

In 8.7.1 and higher, the global sparse matrices can be written to \*.mtx files when the sparse or iterative solvers are used. The index matrices are also written.

### **Element initial strains/stresses**

In 8.7.3 and higher, element initial strains/stresses entered in the STRAIN-FIELD command always act in the global coordinate system.

### **Cyclic plasticity and the truss element**

In 8.7.2 and higher, the plastic-cyclic material model can be used with the truss element.

### **Varying thickness plane stress elements**

In 8.7.3 and higher, plane stress elements can have varying thickness. "Varying thickness" means that the thickness at each element local node can be different. The input for varying thickness plane stress elements is very similar to the input for varying thickness shell elements.

### **Contact surface modification of prescribed loads feature**

In certain problems, it is desirable for contact surfaces to control the prescribed loads acting on those surfaces. For example, there might be a prescribed pressure acting on a contact surface. It might be desirable for this prescribed pressure to be active only when the contact surface is not in contact, and for this prescribed pressure to be inactive when the contact surface is in contact.

The "contact surface modification of prescribed loads" feature (added in 8.7.3) provides this control. This feature is off by default. The feature can be turned on using the PRESSURE-CONTROL parameter of the CONTACT-CONTROL command.

Contact surfaces can modify the following prescribed loads: pressure loads acting on 2D, 3D, plate and shell elements. These loads must be deformation-dependent.

The contact surfaces that can modify prescribed loads are: implicit constraint function with "new" contact surfaces, either 2D or 3D contact, either single or double-sided contact. Only contactor contact surfaces can modify prescribed loads. It is allowed to use symmetric contact so that all contact surfaces are contactors.

The actual applied pressure at a corner node is equal to the prescribed pressure (including the time function multiplier) multiplied by a prescribed load factor. The prescribed load factor is based on the contact pressure at the node, and also on three input values: the critical pressure, the penetration time and the evacuation time. These values are input using new parameters CRITICAL-PRESSURE, TIME-PENETRATION, TIME-EVACUATION of the CONTACT-CONTROL command. The default values of these parameters are zero.

At the beginning of the first time step, the prescribed load factor is 1.0.

At the beginning of the second time step, the prescribed load factor is 1.0 if the contact pressure at the node is less than the critical pressure, and the prescribed load factor is 0.0 if the contact pressure at the node is greater than the critical pressure.

At the beginning of the remaining time steps, the prescribed load factor is increased or decreased according to:

*Contact pressure less than critical pressure.* The prescribed load factor is increased by the value  $(\Delta t)/(\text{penetration time})$ , with a maximum value of prescribed load factor equal to 1. If the penetration time is equal to zero, the prescribed load factor is increased to 1.

*Contact pressure greater than critical pressure.* The prescribed load factor is decreased by the value  $(\Delta t)/(\text{evacuation time})$ , with a minimum value of prescribed load factor equal to 0. If the evacuation time is equal to zero, the prescribed load factor is decreased to 0.

The prescribed load factor is updated at the beginning of the time step, and is unchanged throughout the time step.

It is allowed to set the critical pressure to zero. Then if the contact node is out of contact, the prescribed load factor is increased, and if the contact node is in contact, the prescribed load factor is decreased.

It is allowed to change the critical contact pressure, penetration time and evacuation time in a restart analysis.

The prescribed load factors can be listed using the ADINA-PLOT variable PRESCRIBED\_LOAD\_FACTOR. The prescribed load factors are used in the plots of prescribed pressures.

The following points are important to remember:

a) Contact pressure is force/unit length for 2D planar contact and is force/unit area for 2D axisymmetric and 3D contact.

- b) In symmetric contact, the contact pressure is typically less than the expected value. For example, if two identical blocks are pressed against each other without friction, it is expected that the contact pressure is equal to the compressive stress in the blocks. But if symmetric contact is used in this case, the contact pressure will only be half of the expected value.
- c) The prescribed load factor is only evaluated at corner nodes. The applied pressure at a mid-side node is interpolated from the applied pressures at the corner nodes, regardless of the value of the contact pressure at the mid-side node.
- d) The contact pressure is the normal part of the contact traction vector.
- e) Contact tractions are computed in an approximate way from the contact forces.

### **Contact traction improvements**

In 8.7.3, the contact tractions computed in the implicit constraint function algorithm, using "new" contact surfaces, have been improved. Improvements are made for both 2D and 3D contact surfaces.

This improvement is especially noticeable in 2D analysis for a node in contact surrounded by two nodes not in contact. In this case, the improved traction is double the unimproved traction, and the traction integrated over the contact area is now equal to the contact force.

### **DF-concrete material model**

A new data-fitted concrete material model is added in version 8.7.4. Please see the Theory and Modeling Guide for details.

- Command-line:  
MATERIAL DF-CONCRETE (new command)

## **ADINA CFD & FSI improvements**

### **Sparse solver**

The sparse solver for the Linux 64-bit version now runs 10-20% faster in 8.7.2 and higher.

### **Improvements in turbulence models**

In 8.7.1 and higher, the turbulence models in ADINA-CFD are improved.

## AUI improvements

### Command Window in user interface

Starting in 8.7.1, commands typed into the user interface Command Window are placed into a buffer. To execute a command, press Enter on an empty line, or, to cancel a command, press Esc. Multiline commands are supported. Commands can be copied and pasted into the command line input. The up/down arrow keys can be used to retrieve previous commands.

### Surface functions and geometry faces

Starting in 8.7.1, surface functions are ignored when used for ADINA-CFD element loadings applied to geometry faces.

### Phi model completion

When there are two connected fluid elements from different groups, the number of "uncovered elements faces" reported was incorrect in 8.7.2 and lower. This is fixed in 8.7.3.

### Band plots on picked meshes

When there is more than one meshplot displayed, the "current" meshplot is the one with brackets displayed around it. When plotting bands, the bands are now always (in 8.7.3 and higher) plotted on the current meshplot.

Other results, such as vector plots, reaction plots, etc. also now are always plotted on the current meshplot.

### Band plots -- band table options

BANDTABLE AUTOMATIC ... LOWER UPPER ROUNDING  
BANDTABLE REPEATING ... ROUNDING

New parameters LOWER, UPPER (added in 8.7.1) have the values {AUTOMATIC / NONE / (color name)}, default AUTOMATIC. LOWER controls the color for values less than the lowest value in the band table, UPPER controls the color for values greater than the largest value in the band table. LOWER and UPPER are not used when plotting line contours.

New parameter ROUNDING (added in 8.7.1) has the values (NO / YES), default YES. When ROUNDING=NO, then the min and max band table values are not rounded. The AUI uses either the automatically generated min and max band table values, or the user-entered values.

When BANDTABLE AUTOMATIC ... ROUNDING=NO, the band table is changed slightly as follows. The minimum and maximum values in the band table are always labeled. In addition, as many evenly spaced values as possible are labeled.

### **Band plots -- plots of prescribed quantities**

The following variables (added in 8.7.1) can be plotted as bands on 3D, plate and shell elements (in midsurface depiction).

PRESCRIBED\_PRESSURE, PRESCRIBED\_NORMAL\_TRACTION,  
 PRESCRIBED\_HEATFLUX, PRESCRIBED\_SEEPAGE\_FLUX,  
 PRESCRIBED\_MASSFLUX\_\*, PRESCRIBED\_POREFLOW,  
 PRESCRIBED\_CURRENT\_DENSITY, PRESCRIBED\_PHIFLUX,  
 PRESCRIBED\_EM-FLUX

Notes:

- i) These variables cannot be used by any other command, including ALIAS or RESULTANT.
- ii) These variables plot on elements cut by cutting planes.
- iii) The sign of these variables is correct: positive pressures and normal tractions act into the elements, positive heat flux adds heat to the elements, etc.
- iv) The usual band table options are applicable.
- v) The "close extremes" feature is not implemented for these variables.
- vi) Multiple loads acting on the same element faces are correctly combined.

### **Element vector plots -- vector quantity command**

The new EVECTORQUANTITY command (added in 8.7.1) allows you to define a quantity for vector/tensor plotting using EVECTORPLOT. The syntax of the EVECTORQUANTITY command is

```
EVECTORQUANTITY QUANTITY TYPE V1 ... V6
```

```
QUANTITY                                     [ ' ]
Name of a quantity.
```

```
TYPE                                         [VECTOR]
Type of quantity {VECTOR/SYMTENSOR}.
```

```
V1, ..., V6                                  [ ' ]
```

When TYPE=VECTOR, V1 to V3 are variables giving the X, Y and Z components of the vector. When TYPE=SYMTENSOR, V1 to V6 are variables giving the XX, YY, ZZ, XY, XZ, YZ components of a symmetric tensor. It is allowed to enter 'NONE' for any variable; then zero is substituted for that variable.

Example:

```
EVECTORQUANTITY DISP VECTOR,  
    'X-DISPLACEMENT' 'Y-DISPLACEMENT' 'Z-DISPLACEMENT'  
EVECTORPLOT QUANTITY=DISP // plots displacements as vectors
```

### **Element vector plots -- vector grid site option**

EVECTORGRID SITEOPTION=NODES

With this option (added in 8.7.1), the evector quantity can be plotted at nodes (if possible). Quantities on shell elements can only be plotted on element faces, not within elements and not at nodes.

### **Element vector plots -- vector filter option**

EVECTORRENDERING FILTER

New parameter FILTER (added in 8.7.1) can be ALL (plots all quantities, the default), POSITIVE (plots only positive quantities), NEGATIVE (plots only negative quantities), NONE (plots nothing). FILTER is useful only for tensor quantities. FILTER is ignored for crack / crush quantities.

### **Contact analysis variables**

NODAL\_CONTACT\_OVERLAP:

= -NODAL\_CONTACT\_GAP, if NODAL\_CONTACT\_STATUS > 2, otherwise 0

The purpose of this variable (added in 8.7.1) is to easily list or plot the areas where the overlap is large. This shows if the contact compliance factor is too big or too small.

NODAL\_STICK\_VELOCITY:

= NODAL\_SLIP\_VELOCITY, if NODAL\_CONTACT\_STATUS = 5, otherwise 0.

The purpose of this variable (added in 8.7.1) is to determine the velocity of "sticking" nodes. This is especially important for explicit contact, so that the frictional penalty can be adjusted.

## Graph scaling

Graph scaling is improved in 8.7.3. Axes are allowed to contain more major ticks than before, especially when the axis numbers are smaller than the default value. Also log axes can contain more major ticks than before.

# Modal damping ratio calculations

## Objective

To enable the user to calculate appropriate values of modal damping ratios  $\zeta_i$  in transient modal analysis, based on strain energy proportional damping ratios  $h_i$ . These damping ratios can be either constant over the whole structure, or locally varying between components (element groups). This feature is available in ADINA 8.7.3 and higher.

## Details

In damped modal analysis in ADINA, the following system of equations is solved

$$\ddot{x}_i(t) + 2\omega_i \zeta_i \dot{x}_i(t) + \omega_i^2 x_i(t) = r_i(t) \quad i = 1, 2, \dots, p$$

where  $i$  is the mode number, ranging from 1 to  $p$ ,  $\omega_i$  is the natural frequency and  $\zeta_i$  is the *modal damping ratio*. The modal equations are all decoupled. For more details, see Section 7.3.2 of the ADINA-TMG, or Section 9.3.3 of the Finite Element Procedures book.

If the user requires a different form of damping, such as strain energy damping, then that damping form must be converted to “equivalent” modal damping ratios. A standard damped modal analysis is then performed.

Two types of strain energy proportional damping are allowed by ADINA. In the first type, *energy damping ratios* applicable to the whole structure are input. In the second type, the energy damping ratios are allowed to vary between different element groups.

### Global strain energy damping

The energy damping ratio  $h_i$  for strain energy proportional damping is defined for each mode  $i$  as

$$h_i = \frac{\Delta E_i}{E_i} \quad (1)$$

where  $\Delta E_i$  is the energy dissipated in the  $i$ -th mode, and  $E_i$  is the total strain energy for that mode. We assume that this energy damping ratio provides the energy dissipated per unit

radian.

The total strain energy for mode  $i$  in modal analysis is proportional to the projected stiffness matrix:

$$E_i = \frac{1}{2} \boldsymbol{\varphi}_i^T \mathbf{K} \boldsymbol{\varphi}_i = \frac{1}{2} \omega_i^2 ; \quad \boldsymbol{\varphi}_i^T \mathbf{M} \boldsymbol{\varphi}_i = 1 \quad (2)$$

where  $\boldsymbol{\varphi}_i$  is mode shape  $i$  based on an undamped system and  $\mathbf{K}$  and  $\mathbf{M}$  are the global stiffness and mass matrices.

#### Local strain energy damping

If the strain energy damping ratios are not uniform throughout the structure, ADINA will allow each sub-structure or component (represented as element group in ADINA) to have its own *local energy damping ratios*

$$h_{i|j} = \frac{\Delta E_{i|j}}{E_{i|j}} \quad (3)$$

where  $\Delta E_{ij}$  and  $E_{ij}$  are the dissipated and total strain energy for mode  $i$  for element group  $j$ .

The strain energies in Eqn. (3) can be expressed as

$$E_{i|j} = \frac{1}{2} \boldsymbol{\varphi}_i^T \mathbf{K}_j \boldsymbol{\varphi}_i$$

where  $\boldsymbol{\varphi}_i$  is undamped mode shape  $i$  and  $\mathbf{K}_j$  is the stiffness matrix for element group  $j$ . In practice, only the subset of  $\boldsymbol{\varphi}_i$  involving nodes in element group  $j$  will be used. Note that the mode shapes of the whole structure are used.

ADINA allows each element group to have its own input of local energy damping ratios  $h_{ij}$  and then calculates the global energy damping ratios using

$$h_i = \frac{\sum_{j=1}^N E_{i|j} h_{i|j}}{\sum_{j=1}^N E_{i|j}}$$

where  $N$  is the number of element groups. This calculation is performed before the damped modal analysis, using the mode shapes

$$h_i = \frac{\sum_{j=1}^N \boldsymbol{\varphi}_i^T \mathbf{K}_j \boldsymbol{\varphi}_i h_{i|j}}{\sum_{j=1}^N \boldsymbol{\varphi}_i^T \mathbf{K}_j \boldsymbol{\varphi}_i} = \frac{1}{\omega_i^2} \sum_{j=1}^N \boldsymbol{\varphi}_i^T \mathbf{K}_j \boldsymbol{\varphi}_i h_{i|j} \quad (4)$$

Once the global strain energy damping ratios  $h_i$  are calculated, the modal damping ratio for each mode is set by ADINA as

$$\xi_i = \frac{h_i}{2} \quad (5)$$

A standard damped modal analysis is then performed.

### Notes

It is the user's responsibility to provide proper values of strain energy damping ratios,  $h_i$  in the case of global parameters, or  $h_{ij}$  in case of local damping.

In the case of local strain energy damping, the AUI will provide means for the user to apply the damping ratios to multiple element groups simultaneously, instead of each element group separately.

Equation (5) is applicable for small damping ratios. It can be derived, for a single degree of freedom model, by comparing the rate of decay resulting from the damping ratio  $\xi_i$ , and that from the strain energy damping ratio  $h_i$ . For viscous damping, the following relationship holds for small amount of damping

$$\xi = \frac{\delta}{\sqrt{\delta^2 + 4\pi^2}} \quad (6)$$

where  $\delta$  is the logarithmic decrement

$$\delta = \ln\left(\frac{x_0}{x_1}\right)$$

which can be expressed in terms of the strain energy damping ratio, as follows

$$\delta = -\frac{1}{2} \ln\left(\frac{E_1}{E_0}\right) = -\frac{1}{2} \ln\left(\frac{E_0(1-h)^{2\pi}}{E_0}\right) = -\pi \ln(1-h) \quad (7)$$

Substituting Eqn. (7) into Eqn. (6) and assuming small values of  $\zeta$  and  $h$ , results in Eqn. (5).

### **AUI input**

Modal damping ratios are calculated if one of the modal analysis options is specified (e.g., MODAL-TRANSIENT, MODAL-PARTICIPATION-FACTORS, etc.) and if the following command is used:

```
MODAL-DAMPING  OPTION=LOCAL  EGROUP= . . .
```

This command can be used for every element group. If EGROUP = 0 is specified then the damping factors are taken for all groups not listed in the MODAL-DAMPING command input. Note that if MODAL-TRANSIENT analysis is specified, then modal damping factors are taken according to Eqn. (5).

Modal damping ratios are listed in the output (.out) file.

### **Reference**

K. Kawashima, H. Nagashima and H. Iwasaki, "Evaluation of Modal Damping Ratio Based on Strain Energy Proportional Damping Method", *Journal of Structural Engineering*, Vol 40A, 1994.

# Available documentation

The following documents are available with the ADINA System. These documents are modified in this release of the ADINA System as described below.

## **ADINA User Interface Command Reference Manual**

**Volume I: ADINA Solids & Structures Model Definition, Report ARD 10-2, June 2010**

**Volume II: ADINA Heat Transfer Model Definition, Report ARD 10-3, June 2010**

**Volume III: ADINA CFD & FSI Model Definition, Report ARD 10-4, June 2010**

**Volume IV: Display Processing, Report ARD 10-5, June 2010**

Updates are made for the new and updated commands.

## **ADINA Primer, Report ARD 10-6, June 2010**

Problem instructions are revised for the ADINA System 8.7.

## **Theory and Modeling Guides**

**Volume I: ADINA Solids & Structures, Report ARD 10-7, June 2010**

**Volume II: ADINA Heat Transfer, Report ARD 10-8, June 2010**

**Volume III: ADINA CFD & FSI, Report ARD 10-9, June 2010**

The new features of the solution programs are described.

## **ADINA Verification Manual, Report ARD 10-10, June 2010**

## **TRANSOR for I-deas Users Guide, Report ARD 10-15, June 2010**

## **Nastran Interface manual**

Describes the Nastran commands that are supported by the AUI.

This page intentionally left blank