

**A**UTOMATIC

**D**YNAMIC

**I**NCREMENTAL

**N**ONLINEAR

**A**NALYSIS

## Release Notes

ADINA 9.4.2

July 2018

ADINA R & D, Inc.

# ADINA System 9.4

## Release Notes

(for version 9.4.2)

July 2018

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. 2018  
July 2018 Printing  
Printed in the USA

---

# New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.4, as compared with ADINA System 9.3.4.

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

### Table of supported platforms, SMP program versions

Platform	Operating system	Fortran compiler
Linux x86_64	2.6.18 and higher, glibc 2.5 and higher, gcc 4.1.2 and higher	Intel ifort 11.1
Linux 86_64, AVX extensions	2.6.32 and higher, glibc 2.12 and higher, gcc 4.4.6 and higher	Intel ifort 14.0.4
Windows x86_64	Windows 7, 8, 8.1, 10	Intel Visual Fortran 14
Windows x86_64, AVX extensions	Windows 7, 8, 8.1, 10	Intel Visual Fortran 14

1) All program versions are 64-bit, using the x86\_64 architecture. The Intel 64 and AMD Opteron implementations of the x86\_64 architecture are supported.

## Table of supported platforms, DMP program versions

Platform	Operating system	Fortran compiler
Linux x86_64	2.6.18 and higher, glibc 2.5 and higher, gcc 4.1.2 and higher	Intel ifort 11.1
Linux 86_64, AVX extensions	2.6.32 and higher, glibc 2.12 and higher, gcc 4.4.6 and higher	Intel ifort 14.0.4

- 1) The DMP program versions are based on OpenMPI version 3.0.0.
- 2) The DMP program version with AVX extensions is newly added for version 9.4.
- 3) The DMP program versions do not support all features of the SMP program versions, see the DMP Users Guide for details.

## Sparse solvers

The sparse solvers have been substantially improved in version 9.4. The solution time required is often reduced by a factor of 4 or more for large models, and the memory required is also reduced. Because the 3D-iterative solver and subspace iteration solver use the sparse solver, the solution time required by these solvers is also reduced.

The sparse solvers also make better use of the underlying CPU architecture, so the improvement in the sparse solvers is greater when using later CPU architectures (for example, the Ivy Bridge or later architectures).

The sparse solvers for the DMP versions are improved in version 9.4.2, see the section “Improvements between 9.4.1 and 9.4.2”.

## ADINA Solids & Structures features

### Enriched subspace iteration method

A novel extension of the Bathe subspace iteration method for the solution of generalized eigenvalue problems in structural dynamics has been presented by K.T. Kim and K.J. Bathe, “The Bathe subspace iteration method enriched by turning vectors”, *Computers & Structures*, 186 (2017), pp 11-21. The method has been implemented into the ADINA program under the name of “Enriched subspace iteration method”.

As a first step, the method uses the standard (Bathe) subspace iteration method to perform one iteration using standard starting vectors. The Enriched subspace method then uses a 10-step procedure described in the above reference, where turning vectors are calculated to achieve fast convergence. After convergence is reached, a Sturm sequence check can be performed.

The default number of iteration vectors,  $q$ , used is  $\max(1.4p, p+8)$ , where  $p$  frequencies and associated mode shapes are to be calculated

The solution time required is often reduced by a factor of 3 or more for large models as compared to the standard subspace method.

➤ Command-line:

FREQUENCIES ... METHOD={.../ENRICHED-SUBSPACE)

### **New robust frequencies within an interval calculations**

The subspace iteration method can be employed to calculate frequencies within an interval. Lower and upper bounds of frequencies need to be specified as well as the maximum number of frequencies ADINA can calculate. The program then checks how many frequencies exist in the specified interval and calculate them all if the number of existing frequencies is smaller or equal to the maximum number of frequencies specified. If the number of existing frequencies is larger than the specified maximum number of frequencies, then the program stops with a message that more frequencies exist than the maximum number provided via input.

### **Other improvements to frequency calculations**

The default number of frequencies to be calculated depends on the type of frequency calculations, as follows:

100 if frequencies are confined to an interval and the subspace method is used.

0 if using the CMS method with Bathe subspace iteration, in which case only the reduced system matrices are calculated in the first phase of CMS.

1 otherwise.

The default number of iterations for the subspace iteration method is increased from 24 to 48.

### **Improvements for SMP**

The CPU time for element assembly is reduced when using SMP with multiple threads. The improvement is made for all element types, but the improvement is greatest when using collapsed 3-D solid elements, in particular for large 4-node tetrahedral models, and models where a master node of a rigid link spider is connected to many slave nodes.

### **Time function scaling of mass and damping matrices**

The mass matrix and damping matrix can be scaled by time functions when low-speed dynamics or implicit dynamics is used. This feature can be used to control the low-speed dynamics stabilization forces. This feature can also be used to ensure that the final converged low-speed dynamics solution satisfies static equilibrium.

➤ Command-line:

```
AUTOMATIC TIME-STEPPING ... LSMASS-TF LFDAMP-TF
```

### **Improvements to lumped mass matrices for 3D solid elements**

The lumped mass matrices for the 5-node pyramid, 13-node pyramid and 14-node pyramid 3D solid elements are improved. In version 9.3, the element mass is divided evenly between the element nodes, *before* the brick is collapsed into the pyramid, and in version 9.4, the element mass is divided evenly between the element nodes, *after* the brick is collapsed into the pyramid.

For example, in version 9.3, the 5-node pyramid is treated as a collapsed 8-node brick, in which the apex of the pyramid is formed by collapsing the top four nodes of the brick, therefore half of the element mass is assigned to the apex node and one-eighth of the element mass is assigned to each of the base nodes. In version 9.4, the apex node and each of the base nodes is assigned one-fifth of the element mass.

### **Improvements to Noh-Bathe method for explicit time integration**

The load magnitude at the mid-step can be chosen to be either the average load for the step (as in 9.3) or can be directly evaluated from the time functions.

➤ Command-line:

```
ANALYSIS DYNAMIC-DIRECT-INTEGRATION ...  
MIDLOAD={MEAN/TIMEFUNCTION}
```

### **RBE3 elements**

ADINA Structures now supports RBE3 elements. An RBE3 element constrains the motion of a slave node to the weighted average of the motion of a set of master nodes.

➤ Command-line:

```
RBE3
```

### **Improvements to connector elements**

Connector elements can reference multilinear matrices. In this case, the stiffness and/or damping matrices are directly entered and are multilinear functions of relative displacement, relative velocity, elastic force, damping force or temperature.

➤ Command-line:

```
EGROUP CONNECTOR ... SUBTYPE={.../MATRIX-MULTILINEAR)  
CONN-PROP MATRIX-MULTLINEAR  
MATRIX-NL-K  
MATRIX-NL-C  
MATRIX-K  
MATRIX-C
```

## **Improvements to element birth-death**

Version 9.4 has a number of improvements related to element birth-death. Therefore models with birth-death might give different results in version 9.4 as compared to version 9.3, especially if the element with birth-death contribute to the mass matrix.

One noteworthy change is the following.

In version 9.3, a DOF is marked as “attached to a dead element” if the DOF is attached to a dead element. If the DOF has a zero pivot in the stiffness matrix, a large value is assigned to the pivot and the equation solution continues.

However, this approach can lead to difficulties if the DOF is attached both to a dead element and an active element. In this case, if the DOF has a zero pivot in the stiffness matrix (for example, due to rigid-body motions of the assemblage), a large value is still assigned to the pivot. This large value removes the (physically present) rigid body motion.

In version 9.4, a DOF is marked as “attached to a dead element” if the DOF is not attached to any active elements. So in the above example, the DOF is unmarked and the zero pivot is printed.

Therefore certain models that had no zero pivots in version 9.3 might have zero pivots in version 9.4. The workaround is to identify the source of the zero pivots (for example, 6 DOF shell elements with drilling stiffness), and to change the model to remove the zero pivots.

## **Thermo-plastic-cyclic material model**

The thermo-plastic-cyclic material model is a new material model in version 9.4.

The thermo-plastic-cyclic material model is a temperature-dependent version of the plastic-cyclic material model already available. All material constants are temperature-dependent. The thermo-plastic-cyclic material model can be used with truss, 2-D solid, 3-D solid, beam, shell and 3-D shell elements.

- Command-line:  
MATERIAL THERMO-PLASTIC-CYCLIC

## **Improvement to thermo-plasticity with kinematic hardening**

The back stress temperature correction implemented in version 9.3 is now used by default in the thermo-plastic material models with kinematic hardening.

- Command-line:  
KINEMATICS BACKSTRESS-CORRECTION={YES}



### **Improvements to Bergstrom-Boyce material model**

The material parameters can be calibrated either in accordance to the Bergstrom-Boyce model defined in the paper by Bergström and Boyce (1998), or in accordance with the MCalibration software developed by Veryst Engineering.

An additional option is available for entering the flow resistance (viscous deformation rate constant for network B).

➤ **Command-line:**

```
RUBBER-VISCOELASTIC BERGSTROM-BOYCE CALIBRATION TAUBASE
```

### **Improvements for plasticity, creep and viscoelastic material models**

For 2D solid, 3D solid and 3D-shell elements, the large strain kinematics are now used by default when large displacement kinematics are selected. This improvement is made by the addition of a new parameter value in the `KINEMATICS STRAINS` command. When `KINEMATICS STRAINS=DEFAULT` and `EGROUP STRAINS=DEFAULT`, small strain kinematics are used when small displacement kinematics are selected, and large strain kinematics are used when large displacement kinematics are selected.

This change is made because, even when the strains are relatively small (~2%), the results are more accurate and the convergence is more robust when large strain kinematics are used.

For the other shell elements that allow large strain kinematics (single-layer MITC3, MITC4, MITC6, MITC9, MITC16 elements), when `KINEMATICS STRAINS=DEFAULT` and `EGROUP STRAINS=DEFAULT`, small strain kinematics are used for both small and large displacement kinematics.

As before, the large strain kinematics can be explicitly selected or deselected using `KINEMATICS STRAINS=(SMALL or LARGE)`, or `EGROUP STRAINS=(SMALL or LARGE)`.

This change has no effect if `KINEMATICS STRAINS=(SMALL or LARGE)`, or `EGROUP STRAINS=(SMALL or LARGE)` are explicitly specified in the model input.

➤ **Command-line:**

```
KINEMATICS STRAINS={.../DEFAULT}
```

## Improvements to output of results for large strain plasticity models

The Hencky strains can now be output (and is the default output) for the following plasticity models, when used with large strain kinematics.

MATERIAL ANAND  
MATERIAL CREEP  
MATERIAL CREEP-IRRADIATION  
MATERIAL CREEP-VARIABLE  
MATERIAL GURSON-PLASTIC  
MATERIAL MROZ-BILINEAR  
MATERIAL MULTILINEAR-PLASTIC-CREEP  
MATERIAL MULTILINEAR-PLASTIC-CREEP-VARIABLE  
MATERIAL PLASTIC-BILINEAR  
MATERIAL PLASTIC-CREEP  
MATERIAL PLASTIC-CREEP-VARIABLE  
MATERIAL PLASTIC-MULTILINEAR  
MATERIAL THERMO-PLASTIC

- Command-line:  
PORTHOLE ELEM-RESULT={.../STRAIN}

## Improvements for contact

### *Nodal sliding distance*

The nodal sliding distance is now computed for frictional contact. The nodal sliding distance is accessed using variable NODAL\_SLIDING\_DISTANCE.

### *Friction delay*

In version 9.4, there are three friction delay options:

- |                     |                                                                                                                                                              |
|---------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <i>No</i>           | No friction delay (the default)                                                                                                                              |
| <i>Frictionless</i> | Frictionless conditions are assumed in a solution step in which contact is established. This option corresponds to the friction delay option in version 9.3. |
| <i>Stabilized</i>   | Stabilized frictional conditions are assumed in a solution step in which contact is established. Stabilized friction delay is introduced in version 9.4.     |

In addition, a global flag is added to the CONTACT-CONTROL command so that the friction delay option can be toggled on and off for the entire model.

➤ **Command-line:**

```
CONTACT-CONTROL FRIC-DELAY  
CGROUP FRIC-DELAY={NO/FRICTIONLESS/STABILIZED/DEFAULT}
```

*Contact slip velocity loading*

The sign convention for contact slip velocity loading onto 3D contact surfaces has been reversed in version 9.4. This reversal is done so that the contact slip velocity loading works the same way for both 2D and 3D contact surfaces; contact slip velocity loading is always added to the nodal velocities to obtain the total contact surface velocity.

Contact slip velocity loadings can no longer be applied in explicit analysis. In previous versions, any contact slip velocity loading is ignored in explicit analysis.

The description of contact slip velocity loading is improved in the ADINA Structures Theory and Modeling Guide, Section 5.11.

**Improvements for element group based mass-proportional loading**

Mass proportional loads can now be applied to a portion of a model (i.e. group-based mass proportional loads) in addition to the existing model-based mass proportional loads. Any number of element groups can be included in the group-based mass proportional load calculations. Nodal forces for group-based mass proportional loads are calculated in the same way as for model-based mass proportional loads.

**Improvements for solution accuracy indicators**

The solution accuracy indicators have been improved, as follows:

An increment of work solution accuracy indicator and a force solution accuracy indicator are now printed.

The same reference is now used in the solution accuracy indicators for both the time steps and bolt steps.

The solution accuracy indicators are now printed for linear and nonlinear implicit dynamics, and linear static analysis with stiffness-stabilization or shell drilling stiffness.

A warning message is now printed to the message file if the stiffness-stabilization indicator or the shell drilling indicator are greater than 5% of the external loading.

**Improvements to energy calculations**

The following improvements are made to energy calculations:

Energy calculations for 3D shell elements

- Energy calculations for iso-beam elements
- Energy calculations for truss elements
- Energy calculations for spring elements
- Energy calculations for thermal strain energy
- Energy calculations for creep strain energy
- Energy calculations for thermal-plastic-cyclic material

### **Improvements for OP2 file creation**

The OP2 file is now saved after each converged time step, such that the OP2 result file can be read during the solution.

Modal participation factor analysis results are now saved to the OP2 result file.

Mode superposition analysis results are now saved to the OP2 result file.

OP2 result files can now be saved for models that use the ANALYSIS-SWITCH feature. The solid solution results in an FSI analysis are now saved to the OP2 result file.

For models in which the OP2 result file is to be saved, the AUI now checks on data file generation that the element label numbers are unique for all elements in the model.

### **Improvements for I-deas universal result file creation**

Frequency analysis results are now saved to the I-deas universal result file.

Modal participation factor analysis results are now saved to the I-deas universal result file.

The solid solution results in an FSI analysis are now saved to the I-deas universal result file.

### **EnSight files**

EnSight files can be created by ADINA Structures. Multiple EnSight files can be combined into a single .case file. It is possible to save both the EnSight files and the porthole file in the same solution run. Either a single EnSight file or one EnSight file per time step can be created.

EnSight files are saved to a separate folder in the working directory.

The results to be saved to the EnSight file can now be selected using RESULTS-ELEMENT.

Multiple EnSight case files can now be merged or copied into a single case file. This feature is useful for restart runs.

➤ **Command-line:**

```
ENSIGHT-CASEFILE  
MASTER RESULTS={.../ENSIGHT/ENSIGHT+PORT}  
RESULTS-ENSIGHT  
ENSIGHT-CASEFILE
```

## **ADINA Thermal & TMC features**

### **Improvements to radiosity analysis**

The thermal system of equations is now solved using the sparse solver.

The radiosity heat flux terms in the tangent conductivity matrix can be included using a scaling factor (0.0 = no radiosity heat flux terms, 1.0 = full radiosity heat flux terms).

A steady-state radiosity analysis can now be solved.

A more accurate radiosity calculation for simple axisymmetric models, e.g. concentric spheres, is now included. This option can be selected using RGROUP RADIATION2 ... AXISYMPLE=YES. In addition, the integration order in the circumferential direction can now be chosen to be 48.

➤ **Command-line:**

```
RGROUP RADIATION2 ... CINT AXISYMPLE  
TMC-CONTROL RAD-FACT
```

### **Membrane conduction elements**

Membrane conduction elements are now supported.

### **Improvements to linear thermal analysis**

For linear thermal problems, a linear thermal analysis is performed in one-way TMC and iterative TMC analysis.

In version 9.3, if the structural problem is nonlinear, a nonlinear thermal analysis was always performed, even if the thermal problem is linear.

## **ADINA CFD & FSI features**

### **Reynolds-equation model**

The Reynolds Equation (RE) is used to govern the pressure distribution of thin viscous fluid films in lubrication theory. It has been implemented for FCBI-C elements in version 9.4. The RE is seamlessly integrated into ADINA-CFD and can be used for (but not limited to) laminar

and turbulence flows and coupled with ADINA in FSI computation.

ADINA-CFD supports the RE for both smooth and rough walls. The Patir & Cheng model (see references below) has been implemented for determining effects of roughness on lubrication.

Ref. Nadir Patir and H. S. Cheng, “An Average Flow Model for Determining Effects of Three-Dimensional Roughness on Partial Hydrodynamic Lubrication”, *Journal of Lubrication Technology*, 100 (1978) 14-17.

Ref. Nadir Patir and H. S. Cheng, “Application of Average Flow Model to Lubrication Between Rough Sliding Surfaces”, *Journal of Lubrication Technology*, 101 (1979) 220-229.

The RE is active in specified element groups and at the locations where the boundary distance is less than the specified close/open values defined via the BOUNDARY-DISTANCE command.

It is worth noting that RE is much faster, cheaper and more accurate than the classical Navier-Stokes equation solution when applied to thin fluid film.

It is also worth noting that, although the velocity solution is the boundary-to-boundary averaged quantity, the shear effects in the thin film are still included in the thermal solution, turbulence variables, etc., as well as included in all solution output files (porthole file and graphs \*.ite file). The boundary distance that plays an important role in the RE can be saved as an element result.

➤ Command-line:

```
FCTRL REYNOLDS-EQUATION
RESULTS-ELEMENT ... BOUNDARY-DISTANCE
```

### Universal barotropic cavitation

The Universal Barotropic Cavitation (UBC) is a simple and effective cavitation model. It has been implemented for the FCBI-C element in version 9.4. It can be used in the incompressible flow module of ADINA-CFD (although the two phase flow in essence is compressible flow).

It is worth noting that this model can be used without activating the VOF and/or heat transfer formulations that are usually required in the cavitation model. The UBC model is frequently used together with RE (as well as other fluid formulations) when cavitation occurs.

➤ Command-line:

```
FCTRL BAROTROPIC-CAVITATION
EGROUP TWODFLUID ... MAT2 FCTRL1 FCTRL2
EGROUP THREEDFLUID ... MAT2 FCTRL1 FCTRL2
```

## **Location domains**

Location domains are lists of nodes, elements, boundary condition label numbers, internal faces or external faces, that can be used with solution graphs.

- Command-line:  
LOCATION

## **Solution graphs**

Solution graphs of convergence and solution variables can be created for FCBI-C element results. The solution graphs are saved to the solution monitor (\*.ite) file, and can be viewed during runtime and after the solution is finished using the ADINA Graph Viewer. This feature can be used to graphically monitor convergence and create graphs of solution results.

There are more than one hundred solution variables currently available in version 9.4. The variables are available at nodes, element, internal/boundary faces and/or in the whole computational domain. Some of the solution variables are

- Equation and variable convergence history for all active independent variables
- Memory usage history
- Mass rate history at inlet and outlet boundaries
- Any active solution variable history
- Shear stress, stress, reaction force, etc. on boundary face
- Heat flux and heat flow rate on boundary face
- Torque vector and its magnitude on boundary face
- Shear velocity and y-plus history in turbulence flow
- Gap status history

- Command-line:  
GRAPH

## **Adaptive meshing using SAM**

An additional parameter DISTANCE is implemented in the ADP-CONTROL command. When DISTANCE = 0 (the default), parameters DX, DY, DZ determine both the direction and magnitude for swept adaptive meshing. When DISTANCE > 0, parameters DX, DY, DZ determine only the direction and parameter DISTANCE determines the magnitude.

- Command-line:  
ADP-CONTROL ... DISTANCE

## **EnSight files**

It is possible to save both the EnSight files and the porthole file in the same solution run. Either a single EnSight file or one EnSight file per time step can be created.

EnSight files are saved to a separate folder in the working directory.

The fluid solution results to be saved to the EnSight file can now be selected using NODESAVE-STEPS.

Additional solid and fluid solution results can now be saved to the EnSight file.

Multiple EnSight case files can now be merged or copied into a single case file. This is useful for restart runs and runs with SAM.

➤ **Command-line:**

```
MASTER
RESULTS={ PORTHOLE/ENSIGHT/ENSIGHT+PORT/NEU/NEU+PORT }
RESULTS-ENSIGHT
ENSIGHT-CASEFILE
```

### **Improvements to saving results**

The boundary distance variable can be saved for conditional loading and Reynolds fluid elements.

➤ **Command-line:**

```
RESULTS-ELEMENT ... BOUNDARY_DISTANCE
```

## **AUI features**

### **User interface improvements**

#### **New icons**

The following new icons are added:

- YZ View
- XY View
- XZ View
- Mesh Element Spider

#### **Improvements to the Model Tree**

##### *Improvements to zones and zone groups*

Right-click **Display** of a zone now displays the zone without re-zooming the display.

Right-click **Hide** of a zone is now supported to remove that zone from the display.



Right-click `Rename` is now supported for `ZZ_MULTI-ZONE` and for zones that are currently displayed.

The automatically created zones are now organized into zone groups.

The Model Tree display of zones and zone groups is improved.

#### *Improvements to categories*

Right-click `Display` and `Hide` is now supported for all Model Tree categories to display and hide that category, respectively. This can be used, for example, the display or hide all entities with a given material model, etc.

The Model Tree labels for node sets, element sets, element-edges sets, and element-face sets are now consistent with the set labels. For example, node set 5 is now labeled 5 in the Model Tree.

Right-click `Rename` is now supported for node sets, element sets, element-edges sets, and element-face sets.

The display of the loads and boundary conditions in the Model Tree is now improved.

Right-click `Rename` is now supported for loads and boundary conditions.

Model Tree lists is now limited to 10,000 items. Beyond the limit, the Model Tree summarizes the list.

#### **Improvements to zones and zone groups**

The following right-click options are available for defining zones: `Adjacent`, `Attached`, `Auto-Chain`, `Edge angle [20]`, `Invert`, `Attach to Nodes`.

Mode Tree items can now be picked during zone definition. For example, an element group can be picked in the Model Tree to append that element group to the zone definition.

#### **Improvements for picking**

##### *Improvements to picking geometry*

The following right-click options are available for picking lines and edges: `Adjacent`, `Attached`, `Auto-Chain`, `Edge angle [20]`, `Invert`.

The following right-click options are available for picking element-edges: `Adjacent`, `Attached`, `Auto-Chain`, `Edge angle [20]`, `Invert`.

The right-click picking options only act on the entities displayed in the current mesh plot.

### **Improvements to subordinate dialog boxes in Linux**

Subordinate dialog boxes in Linux now open in front of the parent dialog box.

### **Improvements to the message window**

The message window is automatically cleared when a new database is created.

### **Improvements to the solution graph view**

A new graph viewer, ADVIEW, is supported that reads the following file formats:

ADINA Structures equilibrium iteration files (\*.ite).

ADINA Structures energy files (\*.egy).

ADINA CFD solution files (\*.ite).

Right-click display options are supported in the graph viewer.

### **Improvements to the solution window**

The size of the solution window can be resized by dragging the sides of the window.

### **Improvements to the dialog boxes in Linux**

Clicking on the [x] button on the top right corner of the dialog box closes the dialog box.

### **In-line and block comments in the command-line parser**

In-line comments can now be included using //, as in this example:

```
LINE STRAIGHT 1 P1=2 P2=3 // Top of model
```

Block comments can now be included using /\* \*/ , as in this example:

```
COORDINATES POINTS  
CLEAR  
1 1.0 0.0 0.0  
/* -----  
2 0.0 1.0 0.0  
3 1.0 0.0 0.0  
----- */  
2 0.0 2.0 0.0  
3 2.0 0.0 0.0
```

## **Improvements to command file writing**

The COLORZONE command is now written to the command file.  
Zone groups are now written to the command file.

## **Model definition**

### **Efficiency improvements**

The efficiency of importing Nastran models is improved.

The efficiency of applying fixities is improved.

### **Improvements to extruded lines, surfaces and volumes**

Extruded lines, surfaces and volumes can be defined in terms of the distance and direction of the extrusion.

➤ **Command-line:**

```
LINE EXTRUDED ... DISTANCE  
SURFACE EXTRUDED ... DISTANCE  
VOLUME EXTRUDED ... DISTANCE
```

### **Improvements to Parasolid geometry engine**

The Parasolid geometry engine kernel is now based on Parasolid V27.1.

As a result, free-form meshes based on the Parasolid geometry engine might be different in version 9.4 than in previous versions.

### **Improvements to geometry tolerances**

The tolerances used in geometry and meshing commands are improved. For each tolerance, if “0” is specified, then the tolerance distance is specified by the TOLERANCES GEOMETRIC command.

### **Improvements for entities assigned to geometry points**

There are several types of entities can be assigned to geometry points, for example, fixities, skew systems, concentrated masses and dampers, and rigid links. The entire list of entities is too long to give here.

In version 9.3, the entity is either applied to all nodes close to the point (within a tolerance), or to the node that was chronologically last attached to the point, depending on the type of entity. In version 9.4, by default, the entity is applied to all nodes attached to the point. The default can be changed in order to obtain the 9.3 behavior.

- Command-line:  
CONTROL . . . APPLYPOINT={ATTACHED/TOLERANCE}

### **Improvements for geometry deletion**

When geometry (points, lines, edges, faces, etc.) is deleted, the rigid links, constraints, concentrated masses, concentrated dampers, springs, alignment elements and connector elements associated with the geometry can optionally be deleted.

- Command-line:  
MISC-OPT DELETE-REF={NO/YES}

### **Improvements for element edge-set, element face-set and element set definitions**

Element edge-sets, element face-sets and element sets can be defined in terms of node-sets.

Element edge-sets of shell elements can now be defined using in terms of nodes, points, auto-chaining, attached, and adjacent.

- Command-line:  
ELEDGESET . . . OPTION={.../NODESET}  
ELFACESET . . . OPTION={.../NODESET}  
ELEMENTSET . . . OPTION={.../NODESET}

### **Improvements for node set definitions**

Node sets can be defined in terms of nearby geometry points

- Command-line:  
NODESET . . . OPTION={.../NEARBY-POINT}

### **Improvements to deleting a range of entities**

When deleting a range of entities using the FIRST and LAST options, all entities in the specified range are deleted, irrespective of whether FIRST or LAST entity exists in the model or not.

### **Improvements to element deletion**

An additional option is available that controls which nodes are deleted when elements and element groups are deleted. When NODES or NODE-DELETE=UNUSED, nodes are deleted only if they would be unused after the specified elements are deleted.

- Command-line:  
DELETE EGROUP ALL NODES=UNUSED  
DELETE ENODES NODE-DELETE=UNUSED  
ELDELETE . . . NODE-DELETE=UNUSED

### **Improvements to empty element groups**

On data file generation, if an element group is empty, the AUI issues a warning message file on data file generation but continues. In version 9.3, data file generation was stopped if an element group was empty.

### **Improvements for contact surface definitions**

Contact surfaces can be generated for all contact surfaces in a contact group, or for all contact groups. Contact surfaces can be generated using mixed meshing (the default).

- Command-line:  
CSURFACE ... TYPE  
CSURFACE ... MESHING={V93/MIXED}

### **Improvements to deletion of contact meshes**

Additional options are available for the deletion of contact meshes. The contact meshes can be deleted from a contact surface, from all contact surfaces in a contact group, or from the entire model, using a single command.

- Command-line:  
CSDELETE TWO-D ... DTYPE  
CSDELETE THREE-D ... DTYPE

### **Improvements to boundary condition definitions**

A description can be added to all boundary condition definitions

- Command-line:  
BOUNDARY-CONDITION ... DESCRIPTION

### **Improvements to spring element and axisymmetric truss definitions**

The element group number can be specified in the spring element definition and axisymmetric truss element definition commands.

- Command-line:  
SPRING POINTS ... GROUP  
SPRING LINES ... GROUP  
SPRING NODESETS ... GROUP  
SPRING-6DOF POINTS ... GROUP  
SPRING-6DOF LINES ... GROUP  
SPRING-6DOF NODESETS ... GROUP  
TRUSS-POINTS ... GROUP  
TRUSS-LINES ... GROUP

### Improvements to rigid link definitions

Rigid links can now be defined using volumes and bodies

Any combination of slave and master entity types is now allowed.

- Command-line:  
RIGIDLINK ... MASTERTYPE={.../VOLUME/BODY}

### Improvement to gluemesh definitions

Gluemeshes can be applied between volumes and bodies.

- Command-line:  
GLUEMESH stype<sub>i</sub>=VOLUME

### Improvements to cyclic boundary definitions

The list of slave and master cyclic boundaries can now be order independent. That is, the nth entry of the slave list no longer needs to correspond to the nth list entry of the master list.

Cyclic boundaries can be defined using element-edge sets and element-face sets.

The dialog box for cyclic boundary definitions is improved.

- Command-line:  
CYCLICBOUNDARY ELEDGESET  
CYCLICBOUNDARY ELFACESET

### Improvements to specification of active dofs

Active degrees of freedom of the reduced model for SIMPACK analysis (or boundary degrees of freedom for the CMS method) can be specified using nodesets.

- Command-line:  
DOF-ACTIVE TYPE={NODE/NODESET}

### Improvements in coincidence checking for mesh generation commands

Nodal coincidence checking can be done on all boundary nodes of the meshed body, against all nodes currently displayed in the mesh plot MESHLOT00001.

- Command-line:  
BODY REVOLVED ... NCOINCIDE={.../DISPLAY}  
BODY SWEEP ... NCOINCIDE={.../DISPLAY}  
GLOFTED ... NCOINCIDE={.../DISPLAY}  
GSURFACE ... NCOINCIDE={.../DISPLAY}

```
GVOLUME ... NCOINCIDE={.../DISPLAY}  
GFACE ... NCOINCIDE={.../DISPLAY}  
GBODY ... NCOINCIDE={.../DISPLAY}  
GHEXA ... NCOINCIDE={.../DISPLAY}  
BHEXA ... NCOINCIDE={.../DISPLAY}  
COPY-MESH-BODY ... NCOINCIDE={.../DISPLAY}
```

### **Improvements for moving mesh and special boundary conditions definitions**

Moving mesh and special boundary conditions can be specified using geometric entities or finite element entities.

➤ **Command-line:**

```
MASTER ... ALE={.../GEOM-ELEM}
```

### **Improvements for gap boundary condition definitions**

Gap boundary conditions can be defined using a combination of lines and edges (or surfaces and faces).

Gap boundary conditions can be applied to multiple bodies.

➤ **Command-line:**

```
BOUNDARY-CONDITION GAP ... GTYPE={.../TWO-D/THREE-D}
```

### **Improvements for leader-follower definitions**

The default cone angle for all leader-followers can be specified. Leader-following pairs can be defined using lines and edges, using surface and faces, using points, or using nodes.

➤ **Command-line:**

```
LEADER-FOLLOWER ... ANGLE  
LFOLLOWER LINE-EDGE  
LFOLLOWER SURFACE-FACE  
LFOLLOWER POINT  
LFOLLOWER NODE
```

### **Improvements to wall boundary condition definitions**

When defining moving mesh and special boundary conditions using element entities, the wall boundary condition can be applied to all external boundaries where a velocity or pressure boundary condition has not been applied. This option can be used to change the default boundary condition to a wall. If this option is not used, the default boundary condition is zero traction.

➤ **Command-line:**

```
BOUNDARY-CONDITION WALL ... ALL-EXT=FREE
```

## Improvements for body meshing

There is an option to use mixed meshing. In mixed meshing, mapped meshing is employed when possible, otherwise free-form meshing is employed. By default, mixed meshing is used.

- Command-line:  
GBODY ... MESHING={.../MIXED}

## Improvements to boundary layer meshing

A quadrilateral boundary layer mesh with triangular elements outside the boundary layer can be generated by using GFACE ... NODES=3, 6, or 7.

A prismatic boundary layer mesh with tetrahedral elements outside the boundary layer can be generated by using GBODY ... NODES=4, 10, or 11.

The boundary layer thickness is now allowed to be larger than the radius of curvature of the body edge or body face.

The boundary layer shape can be controlled using a shape distortion parameter. The lower the shape distortion threshold, the better the quality of the boundary layer elements.

- Command-line:  
GFACE ... BLSHAPE  
GBODY ... BLSAHPPE

## Improvements to the placement of midside and midface nodes

A default option is now available for the placement of midside and midface nodes. This default option automatically selects the optimal setting for the placement of these nodes.

- Command-line:  
GFACE/GBODY ... MIDNODES={.../DEFAULT}  
GBODY ... MIDFACENODES={.../DEFAULT}

## Changes to meshing defaults

When meshing faces, the default is changed to mixed meshing. In mixed meshing, mapped meshing is employed when possible, otherwise free-form meshing is employed.

When meshing bodies, the default is changed to force all bounding edges of linked faces to have an even number of subdivisions.

- Command-line:  
GFACE ... MESHING=MIXED  
GBODY ... EVEN=LINK



## **Improvements for reference temperature definitions**

There are two options for reference temperature definitions:

The reference temperature is applied to all nodes. The temperature of a node is the reference temperature plus any temperature loading applied to the node. (APPLY-REF=ALL)

The reference temperature is applied to only the nodes on which temperature loads are not applied. The temperatures of a node without temperature loading is the reference temperature and the temperature of a node with temperature loading is the temperature loading. (APPLY-REF=FREE) This behavior is the default.

➤ Command-line:

```
TEMPERATURE-REFERENCE . . . APPLY-REF
```

## **Improvements for radiation surface definitions**

Radiation surfaces can be defined using element edge-sets or element face-sets.

➤ Command-line:

```
RADIATION-ELEMSET
```

## **Improvements to checking for duplicate elements during data file generation**

The AUI now checks for duplicate connector elements on data file generation.

The first 50 duplicate elements are printed to the message file.

➤ Command-line:

```
CONTROL ... DUPLICATE
```

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

### **RBE3 elements**

RBE3 elements can be displayed, included in zones and highlighted:

➤ Command-line:

```
HIGHLIGHT  
ZONE
```

### **Geometry points in post-processing**

It is now allowed to define geometry points in post-processing.

Geometry points defined in post-processing can be used in commands that reference geometry points, see below.

- Command-line:  
COORDINATES POINTS

### **Commands that reference geometry points, in post-processing**

In previous versions of the AUI, in order to refer to geometry points, it is necessary to load a pre-processing database (.idb file). As an improvement, it is now possible to define a geometry point in post-processing, then the closest node to the geometry point is chosen, without the need to load an .idb file.

The default is to use pre-processing data if an .idb file is loaded, and to use the closest node if no .idb file is loaded.

- Command-line:  
TRACERAKE GNODES ... POINTOPTION={PREPROCESSING/CLOSEST}  
GNCOMBINATION ... POINTOPTION={PREPROCESSING/CLOSEST}  
GNLINE ... POINTOPTION={PREPROCESSING/CLOSEST}

### **Enumerated variables**

Some variables are enumerated, that is, they have integer values with associated string values (for example for PLASTIC\_FLAG, 1 = elastic, 2 = plastic). In listings of enumerated variables, the string values are now listed by default. The CONTROL command can be used to request listing of integer values.

- Command-line:  
CONTROL ... LISTVARENUM

### **Automatically created zones**

The automatically created zones are created only when the number of automatically created zones is less than a threshold value. This change is made because for large models, when the number of automatically created zones is large, the time and memory required to set up the zones is also very large.

By default, the threshold value is 100.

- Command-line:  
CONTROL ... AUTOZONE={YES/NO} or a threshold number

### **Order of nodes in a listing of local node variables**

In a listing of local node variables, the results within each element can be output either in global node number order (as in version 9.3) or in local node number order.

- Command-line:  
RESULTGRID ... RESULTORDER={LOCAL/GLOBAL}

### **Distance along line segments**

The total distance along line segments described by nodes can be listed or graphed using a command sequence such as

```
NODECOMBINATION TEST
ENTRIES NODE
1
10
2
...
DATAEND
POINTLIST TEST VAR=DISTANCE
```

Variable DISTANCE uses the original nodal coordinates, variable DISTANCE\_POSITION uses the updated coordinates.

### **Listing of modal participation factors**

In an analysis in which the modal participation factors are computed for the applied loads (as in modal superposition analysis), the modal participation factors can be listed using the MPFINFO command.

### **Contact variables**

The NODAL\_SLIP\_VELOCITY variables have been moved to the “Contact” category.

### **Improvements to DUMPLIST command**

DUMPLIST can now dump a range of lines, and in each line, dump a range of character positions.

- Command-line:  
DUMPLIST ... ENDLINE STARTCHAR ENDCHAR

# Improvements between 9.4.0 and 9.4.1

## ADINA Solids & Structures features

### Improvements in initial memory allocation

The initial memory allocation is now much smaller for problems in which there are a large number of elements connected to a single node.

### Improvements in analysis zooming

The 11-node tetrahedral elements can be used in zooming analysis.

## AUI features

### Model display and post-processing

#### Function for computing the change in a variable between two successive solution times

The new function DELTA\_LOAD\_STEP is used to obtain the change in a variable between two successive solution times, as described below.

Suppose that the ADINA solver stores results at times  $t_1, t_2, \dots, t_i, \dots, t_n$  in the porthole file. And suppose that the results are requested at time  $t$  in the AUI. Then the bracketing times are determined as in the table on the next page.

Observe that the bracketing times correspond to a backwards difference when the requested time equals one of the times in the porthole file.

The function DELTA\_LOAD\_STEP(X) returns the difference in the value of variable X evaluated at the two bracketing times.

In particular, the function DELTA\_LOAD\_STEP(TIME) returns the difference of the difference of the two bracketing times.

Therefore the ratio DELTA\_LOAD\_STEP(X)/DELTA\_LOAD\_STEP(TIME) returns the approximate time rate of change of variable X.

The solution times  $t_1, t_2, \dots, t_i, \dots, t_n$  need not be equally spaced.

The variable X need not be stored in the porthole file for all of the solution times  $t_1, t_2, \dots, t_i, \dots, t_n$ .

Requested time	Bracketing times
$t < t_1$	none
$t = t_1$	none
$t_1 < t < t_2$	$t_1, t_2$
$t = t_2$	$t_1, t_2$
$t_2 < t < t_3$	$t_2, t_3$
...	...
$t = t_i$	$t_{i-1}, t_i$
$t_i < t < t_{i+1}$	$t_i, t_{i+1}$
$t = t_{i+1}$	$t_i, t_{i+1}$
...	...
$t_{n-1} < t < t_n$	$t_{n-1}, t_n$
$t = t_n$	$t_{n-1}, t_n$
$t > t_n$	none

The variable X cannot be a constant. The variable X cannot be a resultant.

As a simple example, the resultant

```
RESULTANT STRAIN_RATE,
  'DELTA_LOAD_STEP(<STRAIN-XX>)/DELTA_LOAD_STEP(TIME)'
```

along with the command

```
ZONELIST VAR=STRAIN_RATE
```

can be used to compute the approximate strain rate for all of the solution steps.

## Improvements between 9.4.1 and 9.4.2

### Sparse solvers for the DMP versions

The sparse solvers for the DMP versions have been substantially improved in version 9.4.2. The solution time required and the memory required is reduced. Because the 3D-iterative solver and subspace iteration solver use the sparse solver, the solution time required by these solvers is also reduced.

The sparse solvers also make better use of the underlying CPU architecture, so the improvement in the sparse solvers is greater when using later CPU architectures (for example,

the Ivy Bridge or later architectures).

## ADINA Solids & Structures features

### Improvements to OP2 file creation

It is now possible to specify whether the OP2 file is created after each converged time step in which the results are saved (MASTER OP2SAVE=STEP), or at the end of the entire solution after all time steps have converged (MASTER OP2SAVE=END).

If the OP2 file is created after each converged time step, the OP2 file can be loaded during the solution. However, the solution time might be increased.

By default, the OP2 file is created at the end of the solution.

- Command-line:  
MASTER OP2SAVE

### Improvement for bolt groups

The element group number corresponding to each bolt number can now be listed in pre- and post-processing, using the new BOLTINFO command.

- Command-line:  
BOLTINFO

## AUI features

### Model definition

#### Improvement for defining element face-sets

When defining element face-sets using auto-chaining with the option EXTERNAL=GROUP, only the element faces of the element groups listed in the table input are considered in the auto-chain. In previous versions, all element faces were considered.

# Updates to documentation

## ADINA Handbook

The example on pages 66-67 demonstrating the use of transitional pyramids has been updated to be consistent with the program's behavior. The revised example is:

Command input 2.51 demonstrates the use of transitional pyramids between face-linked mapped hexahedral and free-form tetrahedral meshes.

**Command Input 2.51:** Meshing two face-linked bodies. Body 1 is map meshed using 8 node bricks, and body 2 is free- form meshed. Figure 2.50 shows the resulting mesh.

```
feprogram program=adina
body block name=1 option=centered position=vector,
cx1=0.0 cx2=0.0 cx3=0.0 dx1=1.0 dx2=1.0 dx3=1.0
body block name=2 option=centered position=vector,
cx1=0.0 cx2=1.0 cx3=0.0 dx1=1.0 dx2=1.0 dx3=1.0
*
facelink option=all
*
subdivide body 1 mode=length size=0.1
subdivide body 2 mode=length size=0.1
*
egroup name=1 type=threedsolid
egroup name=2 type=threedsolid
*
gbody 1 group=1 nodes=8
gbody 2 group=2 nodes=4 meshing=free-form pyramids=only
*
boxzone b
-10 0 -10 10 -10 10
modeldepic geometry=no
frame/meshplot zone=b
```

Figure 2.50 shows the results of Command input 2.51. Body 1 has been map meshed with hexahedral elements. Body 2 is face linked (see page 14) to body 1 and is meshed using tetrahedral elements and transitional pyramids.

To generate the red mesh containing mostly tetrahedral elements but transition pyramids on the interface with the hexahedral mesh, as shown, the user must:

1. Specify **NODES = 4** to mesh the body using tetrahedral elements.
2. Invoke free-form meshing using **MESHING = FREE-FORM**.
3. Specify **PYRAMIDS = ONLY**.

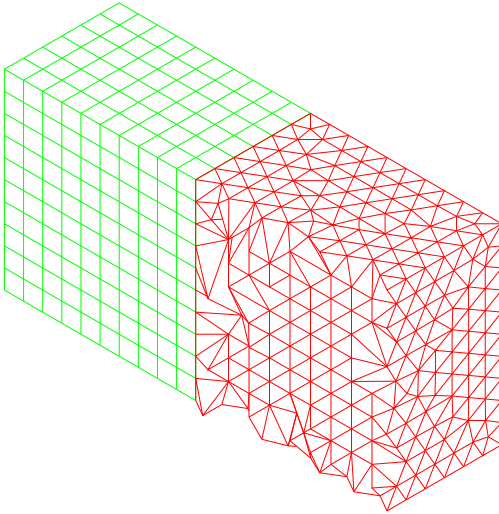


Figure 2.50: Free-form mesh (in red) generated using tetrahedral elements and transitional pyramids.

The **PYRAMIDS = ONLY** option generates transitional elements only on the linked body faces that have quadrilaterals. Transitional pyramids are required for CFD analysis to ensure a compatible mesh. In structural analysis, the **PYRAMIDS = ONLY** option is not required, and tetrahedral elements can be directly attached to the hexahedral elements without transitional pyramids.

## 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.

### Installation Notes

Describes the installation of the ADINA System on your computer. Depending on the platform, the appropriate installation notes in pdf format can be printed or downloaded from <http://www.adina.com>.

### ADINA Handbook

Written as a task-oriented desktop reference, the ADINA Handbook helps users to quickly and effectively leverage ADINA's advanced geometric modeling, meshing, and visualization features.



**ADINA User Interface Command Reference Manual (CRM)**

**Volume I: ADINA Solids & Structures Model Definition**

**Volume II: ADINA Thermal Model Definition**

**Volume III: ADINA CFD & FSI Model Definition**

**Volume IV: ADINA EM Model Definition**

**Volume V: Display Processing**

Updates are made for the new and updated commands.

**ADINA Primer**

Problem instructions are revised for the ADINA System 9.4. Two new primer problems are added.

**Theory and Modeling Guide (TMG)**

**Volume I: ADINA Solids & Structures**

**Volume II: ADINA Thermal**

**Volume III: ADINA CFD&FSI**

**Volume IV: ADINA EM**

The new features of the solution programs are described.

**ADINA Verification Manual**

**ADINA-Nastran Interface Manual**

**TRANSOR for I-DEAS Users Guide**

**TRANSOR for Femap Users Guide**