

**A**UTOMATIC

**D**YNAMIC

**I**NCREMENTAL

**N**ONLINEAR

**A**NALYSIS

## Release Notes

ADINA 9.2.5

October 2016

ADINA R & D, Inc.

# ADINA System 9.2

## Release Notes

(for version 9.2.5)

October 2016

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. 2016  
October 2016 Printing  
Printed in the USA

---

# New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.2, as compared with ADINA System 9.1.3. (For features and improvements added between 9.2 and 9.2.5, see the section “Improvements between 9.2.0 and 9.2.5” 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

### Table of supported 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 XP, Vista, 7, 8, 8.1, 10	Intel Visual Fortran 11
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. The AVX extensions are supported, as described in detail below.

## **AVX extensions**

Recent Intel and AMD CPUs support the AVX (advanced vector extensions) instructions in the x86\_64 instruction set. The ADINA 9.2 executables for the solvers (ADINA Structures, ADINA Thermal, ADINA CFD/EM, ADINA FSI, ADINA TMC) now use the AVX extensions, as described below.

Depending upon the computer and the problem to be solved, the executables that use the AVX extension can run up to 30% more quickly than executables that don't use the AVX extension.

### *Linux versions*

One way to check to see if a computer supports AVX is to search for the string "avx" within the file `/proc/cpuinfo`. If the string is not found, the computer does not support the AVX extensions; if the string is found, the computer supports the AVX extensions.

Two versions of the ADINA executables are supplied, one version that does not use the AVX extensions, and one version that uses the AVX extensions. During installation, if the computer supports AVX, the AVX version is installed by default, and if the computer does not support AVX, the non-AVX version is installed by default. You can override the defaults during installation.

At run-time, the ADINA command scripts check to see if the AVX extensions are supported on the computer. If the AVX extensions are not supported, the non-AVX executables are automatically used by default, and if the AVX extensions are supported, the AVX executables are automatically used by default.

If a computer supports the AVX extension, and you do not want to use the AVX executables, use the `-noavx` flag, for example

```
adi na9.2 -noavx . . .
```

If the solver program is run through the user interface (either through the AUI or using the `ai 9.2` script), a check button is available in the Settings window to run the non-AVX version on a computer that supports AVX.

### *Windows versions*

The installation package contains both the AVX and non-AVX versions. During installation, the computer is checked to see if AVX is supported. If AVX is supported, the AVX version is installed. Otherwise, the non-AVX version is installed.

## ADINA Solids & Structures features

### Improvements to low-speed dynamics

The program can now automatically switch to low-speed dynamics at no convergence, for specific element groups.

The low-speed dynamics inertia factor (LSMASSF) can now be greater than 1.

- Command-line:  
AUTOMATIC TIME-STEPPING LSTIME

### Improvements to energy calculations

The following improvements are made to energy calculations:

- The out-of-core solver is now supported.
- Contact compliance energy is improved.
- Contact damping energy is improved.

### Improvements to 3D plane stress element

The 3D plane stress element (membrane element) can now be initially warped.

### Improvements to mesh glueing

The gluing constraint functions are changed slightly when there are several glued surfaces that intersect. 3D plane stress elements can be used in the glueing of 3D surfaces.

### Change to u/p formulation defaults

The default number of pressure degrees of freedom used in the u/p formulation is changed for the following elements:

*2D plane strain and axisymmetric element:*

5 to 8-node quadrilateral element and the 6-node triangular element: Default number of pressure degrees of freedom is changed from 1 to 3.

*3D solid element:*

9 to 21-node hexahedral or prismatic element, and the 10-node tetrahedral element:  
Default number of pressure degrees of freedom is changed from 1 to 4.

This change is made to make the elements stiffer. The defaults in version 9.1 correspond to elements that are too soft.

Also, please note that the list of material models in which the u/p formulation is used by default has been changed, see below.

### **Change to selection of incompatible modes**

This change involves element groups with materials in which the u/p formulation is used by default, in which the u/p formulation is not explicitly specified, and in which there are no higher-order elements. In this case, if incompatible modes are selected for the element group, either using EGROUP INCOMPATIBLE=YES or using KINEMATICS INCOMPATIBLE=YES, incompatible modes are used and the u/p formulation is not used. Previously the u/p formulation was used and incompatible modes were not used.

### **Bergström-Boyce material model for rubber viscoelasticity**

This material model is now available for 2-D solid elements (plane strain and axisymmetric), and for the 3D-shell element.

Energy dissipated through viscous deformations now causes internal heat generation.

The EOFFSET parameter now has a default value.

➤ Command-line:

RUBBER-VISCOELASTIC BERGSTROM-BOYCE ... GENFAC EOFFSET

### **Three-network model for polymers**

This material model is now available for 2-D solid elements (plane strain and axisymmetric), and for the 3D-shell element.

Energy dissipated through viscous deformations now causes internal heat generation.

➤ Command-line:

MATERIAL BERGSTROM-BOYCE ... GENFAC

### **Improvements to plasticity and creep material models**

*u/p formulation:* The default element formulation has been changed in version 9.2 for the following materials.

ANAND  
CREEP  
CREEP-IRRADIATION  
CREEP-VARIABLE  
MROZ-BILINEAR  
MULTILINEAR-PLASTIC-CREEP

MULTILINEAR-PLASTIC-CREEP-VARIABLE  
PLASTIC-BILINEAR  
PLASTIC-CREEP  
PLASTIC-CREEP-VARIABLE  
PLASTIC-CYCLIC  
PLASTIC-MULTILINEAR  
PLASTIC-ORTHOTROPIC (if elastic properties are isotropic)  
VISCOELASTIC

The u/p (mixed) formulation is now the default when the following material models are used with 2D plane strain, 2D axisymmetric and 3D solid elements. In version 9.1, the displacement-based formulation is the default for these materials.

All of these materials behave incompressibly under inelastic deformations, which is why the u/p formulation should be used with these materials.

#### *Type 2 plasticity*

The “type 2 plasticity” option is now available for the 2D solid elements that use the following material models:

MROZ-BILINEAR  
PLASTIC-BILINEAR  
PLASTIC-MULTILINEAR

#### *Improvement to MASTER CONVERT-SSVAL feature for multilinear plasticity*

The MASTER CONVERT-SSVAL feature converts user-input stress-strain data from engineering to true values. This feature is improved in version 9.2 as follows: the Young’s modulus is also converted, so that the elastic limit remains at the user-specified strain level. More details and a numerical example are given in Section 3.4.1 of the ADINA Structures Theory and Modeling Guide.

#### **Contact traction improvements**

The contact tractions output at the corner nodes of higher-order triangular contact segment have been improved.

#### **Improvements to 3D-bolts**

3D-bolts are now allowed with incompatible meshes in which tetrahedral or prismatic elements with triangular faces are adjacent to hexahedral elements with quadrilateral faces.

## **Saving of results in the porthole file for 2D solid element groups**

The RESULTS-ELEMENT option of saving results is now applicable to 2D solid elements. This feature was already implemented in version 8.9.1 for 3D solid elements.

As a consequence of this update, the results saved in the porthole file are affected.

In the following, the RESULTS-ELEMENT feature is briefly reviewed.

When the RESULTS-ELEMENT feature is used, it is possible to select categories of variables to save in the porthole file. The categories are:

*Stress* (no saving, basic saving, all saving), default all saving  
*Strain* (no saving, all saving), default no saving  
*Inelastic* (no saving, basic saving, all saving), default basic saving  
*Thermal* (no saving, basic saving, all saving), default basic saving  
*Energy* (no saving, all saving), default no saving  
*Electromagnetic* (no saving, all saving), default no saving  
*User-coded variables* (no saving, all saving), default no saving  
*Misc* (no saving, all saving), default no saving

Notice that some categories allow "basic" saving, namely only those results that are commonly used in post-processing are saved.

The category saving feature can be turned off using the command PORTHOLE ... RESULTS-ELEMENT=NO. When PORTHOLE ... RESULTS-ELEMENT=NO, all results are saved at each element integration point. The default is PORTHOLE ... RESULTS-ELEMENT=YES, so that the category saving feature is used by default.

The effect of these improvements is to decrease the number of results written to the porthole file. In particular, by default, the number of results written to the porthole file is decreased. In order to save all results, either set each category in RESULTS-ELEMENT to "all", or use the command PORTHOLE ... RESULTS-ELEMENT=NO.

To set the saving for a given element group, use the command RESULTS-ELEMENT ... GROUP=(group number). To set the saving for all remaining element groups in the model, use the command RESULTS-ELEMENT ... GROUP=0.

➤ Command-line:  
RESULTS-ELEMENT  
PORTHOLE ... RESULTS-ELEMENT

## **Saving of results at corner nodes for 2D solid element groups**

In version 9.2, it is possible to save the results at the element corner nodes, for 2D solid element groups. This feature was already implemented in version 8.9.1 for 3D solid element groups.

The default is to save the results at the element integration points.

- Command-line:  
RESULTS-ELEMENT LOCATION=CORNER

### **Improvements to op2 file writing**

The op2 stress/strain data blocks can optionally be written in “linear” format for both linear and nonlinear analysis. This option is useful when only op2 “linear” data blocks are supported (ie. Fe-Safe, HyperView, etc.).

- Command-line:  
MASTER ... OP2STR

## **ADINA CFD & FSI features**

### **Improvements to material models**

The following real gas models are supported:

- Standard Redlich-Kwong model
- Aungier-Redlich-Kwong model
- Soave-Redlich-Kwong model
- Peng-Robinson model

The ideal gas model can be used with a user-defined specific gas constant. This model can be used when  $C_v$  should vary in the energy equation, but the specific gas constant should not vary in the equation of state.

Temperature, pressure, and time dependent material property curves are now supported for all material models, including the turbulence models. The material property curves can be specified either in tabulated (multilinear) form or in polynomial form.

Material property curves can be defined that are dependent on any two of the following arguments: time, temperature, or pressure for all material models. For example, temperature and pressure dependent material property curves can be entered.

- Command-line:  
STATE-EQUATION  
MATERIAL ... STATE-EQ MCURVE1 MCURVE2 MCURVE3  
MCURVE TABULATED  
MCURVE 2ARGUMENTS  
MCURVE POLYNOMIAL

### **Improvements to loads for FCBI-C elements.**

Loads can be applied onto element groups for FCBI-C elements.

### **Improvements to nodal interpolations for FCBI-C elements**

The nodal results stored on the porthole file for the FCBI-C elements are improved.

### **Improvements to TFSI analysis**

The “Heat Transfer Only” option, MASTER ... TMC-MODEL=HEAT, in the solid model can now be used in TFSI analysis.

### **Improvements to turbulence models**

The search algorithm used to measure the wall distance for turbulent flow calculations in the two-layer and SA turbulence models has been improved.

## **AUI features**

### **User interface improvements**

#### **Improvements to the model tree**

##### *Zone-group feature:*

This feature allows users to group their zones, which is especially useful to organize large models. For example, the user might want to create an “Engine” group that contains all the zones associated with the engine. This group can then be expanded or contracted in the Model Tree using the [+] or [-] buttons, respectively. The user can move zones from one group to another, or multi-select zones and create a new group. In command line format, a ZONE-GROUP command can be used to specify which zones are to be grouped.

- Command-line:  
    ZONE-GROUP

##### *Additional items in the Model Tree:*

The following items are now included in the Model Tree: element face-sets, element edge-sets, element sets and node sets.

### *Automatic highlighting*

When the user clicks on an item in the Model Tree, the item is highlighted in the mesh plot. This is especially useful for zones.

### *Zone colors*

The color of each zone is shown in the Model Tree by coloring the text (this feature is implemented only for the Linux version of the AUI).

### **Improvements to the dialog boxes**

The following dialog boxes have been improved:

*Apply Initial Conditions:* The “Apply Initial Conditions” dialog box has the same form as the “Apply Load” and “Apply Fixity” dialog boxes. That is, the user first defines the initial condition and then specify the geometric entities associated with that individual initial condition only.

*Mesh-Split, Mesh Detach, Mesh Join:* These dialog boxes now have a progress bar.

*Leader-Follower:* The “Label #” field is now automatically populated when the user screen selects the leader-followers. This improves the workflow for large problems that have many tens of leader-followers.

### **Improved resetting of dialog boxes after execution**

For geometry manipulation dialog boxes (i.e., Boolean Operator and Body Modifier operations), and for mesh manipulation dialog boxes (i.e. Split Mesh, Detach Mesh, Join Mesh), and for the node sets, element sets, element-face sets and element-edge sets dialog boxes, these dialog boxes no longer reset to the default settings after execution.

### **Improved interaction between dialog boxes and subordinate dialog boxes**

Many dialog boxes call subordinate dialog boxes using the “...” button. Now the calling dialog box automatically updates with the subordinate label number or name every time the subordinate dialog box is executed (that is, every time the user clicks “Save” or “OK” in the subordinate dialog box). For example,

Click the Change Zone icon.

Click the ... button to the right of the Zone Name field.

In the Define Zone dialog box, define a new zone and click OK.

In the Change Zone dialog box, the Zone Name shows the newly defined zone name.

In this example, the Define Zone dialog box is the subordinate dialog box and the Change Zone dialog box is the calling dialog box.

The following list gives all dialog boxes updated by their subordinate dialog boxes:

- Mesh Faces/Basic→Domain
- Mesh Faces/Basic→Boundary Layer Table
- Mesh Faces/Advanced→Size Function
  
- Mesh Bodies/Basic→Domain
- Mesh Bodies/Basic→Boundary Layer Table
- Mesh Bodies/Tetrahedral→Size Function
  
- Mesh Surface/Options→Domain
  
- Mesh Volume/Nodal Coincidence→Domain
- Define Body (Parasolid)→Block→Coordinate System
- Define Body (Parasolid)→Cylinder→Coordinate System
- Define Body (Parasolid)→Sphere→Coordinate System
- Define Body (Parasolid)→Torus→Coordinate System
- Define Body (Parasolid)→Pipe→Coordinate System
- Define Body (Parasolid)→Cone→Coordinate System
- Define Body (Parasolid)→Prism→Coordinate System
- Define Body (Parasolid)→Transformed→Transformation Label
- Define Body (Parasolid)→Sheet→Coordinate System
  
- Body Sweep→Coordinate System
- Body Revolve→Coordinate System
  
- Define Section Sheet→Coordinate System
  
- Define Volume→Revolved→Coordinate System
- Define Volume→Extruded→Coordinate System
- Define Volume→Transformed→Transformation Label
  
- Change Zone→Zone Name

### **Improvements for picking**

*Right-click menu:* There is now a right-click menu for picking with the following options:

Adjacent  
Attached  
Auto-chain

This menu is available when picking elements (e.g. in “Delete Elements”, “Element Set”, “Element Face-Set”, etc.).

*Screen picking:* When screen picking using the “Cyan” table columns, all items that have been picked are saved to the table, even those that are not in the current mesh plot. In previous versions of ADINA, only picked items displayed in the current mesh plot are saved. This improvement is necessary for commands where coincident faces need to be selected; e.g. MESH-JOIN.

### **Improvements for the Energy View**

The Energy View window can now be dynamically resized.

During post-processing, the Energy View utility can be accessed by choosing Graph→Energy View.

### **Improvements for Fast Graphics**

Custom function keys for dynamics graphics manipulation are implemented. In this way, users familiar with other software packages can manipulate models in the ADINA FGM in the same way as what they are used to.

A space mouse can be used for dynamic graphics manipulation in the ADINA FGM. Using a space mouse for large models is much more effective than using a regular mouse.

## **Model definition**

### **Improvements to constraint and rigid link definitions**

Constraints and rigid links, when defined on the geometry, can also optionally generate nodes. Using this option, it is not necessary to separately generate nodes using, for example, the GPOINT command.

➤ Command-line:

```
CONSTRAINT ... GPOINT NCOINCIDE NCTOLERANCE  
CONSTRAINT-MS ... GPOINT NCOINCIDE NCTOLERANCE  
RIGIDLINK ... GPOINT NCOINCIDE NCTOLERANCE
```

### **Improvements to BODY SWEEP and BODY REVOLVED**

The NCOINCIDE=GROUP option is now supported for BODY SWEEP and BODY REVOLVED. The RATIO, PROGRESSION and CBIAS parameters are now implemented for BODY SWEEP and BODY REVOLVED.

- Command-line:  
BODY SWEEP ...NCOINCIDE=GROUP RATIO PROGRESSION CBIAS  
BODY REVOLVED ...NCOINCIDE=GROUP RATIO PROGRESSION CBIAS

### **Improvements to the SUBDIVIDE EDGE command**

The CBIAS (central bias) parameter is now supported in the SUBDIVIDE EDGE command.

- Command-line:  
SUBDIVIDE EDGE ... CBIAS

### **New POINT commands**

Using the new POINT LINE and POINT EDGE commands, geometry points can be created along a geometry line or geometry edge.

- Command-line:  
POINT LINE  
POINT EDGE

### **LINE EDGE command**

The new LINE EDGE command converts an edge of a body to a geometry line, similar to SURFACE FACE.

- Command-line:  
LINE EDGE

### **Boundary layer meshing**

The following improvements are made to boundary layer meshing:

Boundary layers can be generated on multiple bodies in the execution of one command. In version 9.1, multiple BLTABLE commands and multiple GFACE or GBODY commands must be executed when generating boundary layers on multiple bodies.

Defaults are now set for the progression type, thickness of first layer, number of layers, and total thickness of the boundary layer.

These improvements are made in the new BLTABLE-2D and BLTABLE-3D commands. BLTABLE-2D is to be used with GFACE, and BLTABLE-3D is to be used with GBODY.

- Command-line:  
BLTABLE-2D  
BLTABLE-3D

### **Node deletion**

Nodes that are not referenced by any other entity in the database (such as elements, rigid links, rigid contact surfaces, etc.) can be deleted.

- Command-line:  
NODE-DELETE

### **Node renumbering**

Nodes that are not associated with geometry can be renumbered.

- Command-line:  
NODE-RENUMBER

### **Improvements to element deletion**

Elements that are listed in a table, elements in element groups, and elements listed in element sets can now all be deleted, including those elements associated with geometry.

One example where deleting elements associated with geometry is useful is for problems where the ADINA Structures solution does not converge because of a small number of poor quality sliver elements that do not contribute to the global response of the structure. In this case, the sliver elements can be deleted and the analysis restarted.

In addition, ELDELETE VOLUME and ELDELETE BODY now also delete any 2D solid elements or any shell elements associated with the surfaces and faces of the selected volumes and bodies, respectively.

- Command-line:  
ELDELETE ELEM  
ELDELETE ELSET  
ELDELETE EGROUP

### **Node-set, element-set, element edge-set and element face-set improvements**

Whenever a list is given in the node set, element-face set and element-edge set commands, when the “Method” is switched e.g. “From Geometry Surface/Face”, the new selection can now be appended to the original list. Currently, the original list is overwritten.

- Command-line:
  - NODESET
  - ELEDGESET
  - ELEMENTSET
  - ELFACESET

### **Improvements to mesh generation**

When generating meshes with mid-side nodes using GFACE, the MIDNODES=PROJECT option is now available. Using the MIDNODES=PROJECT feature, 2<sup>nd</sup> order meshes generated using GFACE MESHING=MAPPED are now compatible with 2<sup>nd</sup> order meshes generate using GFACE MESHING=FREE-FORM.

Similarly, using the MIDNODES=PROJECT feature, 2<sup>nd</sup> order meshes generated using GBODY MESHING=MAPPED are now compatible with 2<sup>nd</sup> order meshes generated using GBODY MESHING=FREE-FORM.

- Command-line:
  - GFACE ... MIDNODES=PROJECT

### **Efficiency improvements in the AUI**

When nodes, elements, geometry lines, etc. are deleted, memory is now reclaimed. Therefore the memory requirements are now reduced.

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

### **MPEG file creation (Linux program versions)**

Linux versions of the AUI can create MPEG animation files using the new Save MPEG Movie icon.

### **Post-processing of beam element stresses**

Version 9.2 has new options for the post-processing of beam element stresses:

- 1) Filtered listings, so that only the maximum stresses in the cross-section are listed.
- 2) Plotting of beam element stresses on band plots. In the plots, only the filtered stresses are shown (e.g, the maximum stresses in the cross-sections).

Because only the elastic-plastic beam element outputs stresses, these options for post-processing beam element stresses apply only to the elastic-plastic beam element.

- Command-line:  
RESULTCONTROL ... BEAMCSOPTION BEAMCSIPT

### Changes to short zone names

The CONTROL AUTOZONE feature generates short zone names according to the entities in the model. These short zone names can now be generated either “top-down” or “bottom-up”. As an example, consider contact surface 2 of contact group 3. The “top-down” short zone name is CG3\_CS2 and the “bottom-up” short zone name is “CS2\_CG3”.

Version 9.1 always created bottom-up short zone names. However, bottom-up short zone name do not display in a logical order in the Model Tree. Therefore version 9.2 creates top-down zone names by default. The default can be changed using the CONTROL SZNAME parameter.

- Command-line:  
CONTROL ... SZNAME

### Highlighting of element sets

Element sets can now be highlighted.

### Change to variable name

The output variable name PHYSICAL\_ERROR\_BOUND is changed to EIGENSOLUTION\_ERROR\_MEASURE. This is because the quantity associated with this variable name is not actually an “error bound” in the mathematical sense.

## Improvements between 9.2.0 and 9.2.5

### ADINA Solids & Structures features

#### Using the CMS reduced matrices as general element stiffness / mass matrices

When FREQUENCIES ... AVL-EXCITE=TEXT, ADINA now also outputs the reduced matrices  $\mathbf{K}_r$  and  $\mathbf{M}_r$  in a file with name \*.genl. This \*.genl file can be later imported into the ADINA general element stiffness/mass matrices. This improvement is included in 9.2.1 and higher.

To import the \*.genl file into ADINA, use the MATRIX STIFFNESS and MATRIX MASS commands.

```
MATRIX STIFFNESS ...OPTION=FILE FILE=*.genl  
MATRIX MASS ...OPTION=FILE FILE=*.genl
```

➤ **Command-line:**

```
MATRIX STIFFNESS ... OPTION=FILE  
MATRIX MASS ... OPTION=FILE
```

### **Deletion of temporary files on Windows computers**

In version 9.2.3 and higher, ADINA Structures automatically deletes all temporary files after program execution, even if the solution does not converge.

### **Solution diagnostics**

Some of the solution diagnostics (listed in the .out file when DIAGNOSTICS SOLUTION=DETAILED is specified) have been improved in version 9.2.4 and higher.

### **Frequency analysis with rotational loadings**

Frequency analysis with rotational loadings is no longer possible in version 9.2.5 and higher. This is because the Coriolis acceleration coupling term in the rotational loading is not included in the stiffness matrix (this term causes a nonsymmetric contribution to the stiffness matrix in general.) Because the Coriolis acceleration term is not included, misleading results might be obtained, if frequency analysis with rotational loadings is permitted.

Frequency analysis with centrifugal loading is unchanged in version 9.2.5 and higher, so it is still possible to perform frequency analysis with centrifugal loading.

### **Prescribed displacements with arrival times**

According to documentation, a prescribed displacement with arrival time should be free before the arrival time is reached. However, in 9.2.4 and lower, when the model is not supported in static analysis, so that there are rigid body modes in the analysis, the prescribed displacement is set to zero before the arrival time is reached. This issue is fixed in version 9.2.5 and higher, so that the prescribed displacement remains free before the arrival time is reached. As a result, this type of problem might no longer converge. As a workaround, make sure that the model is properly supported, or use low-speed dynamics or stiffness stabilization.

## AUI features

### User interface improvements

The following changes are made for versions 9.2.2 and higher:

In the Define Cross-Section dialog box, Type=General, the text “Moment of Inertia” is changed to “Area Moment of Inertia”, and the component labels “s”, “t” are changed to “ss”, “tt”.

In the Solution menu, the choice “Steered Adaptive” is changed to “Run Steered Adaptive”

In the Define Element Set dialog box, in the Method drop-down list, the choices “Attached to Elements” and “Adjacent to Elements” are changed to “Attached” and “Adjacent”.

When 3D flow assumptions are used, the [+] button in the Mesh Faces dialog box automatically creates a 2D fluid “Mesh Only” element group.

The following changes are made for versions 9.2.3 and higher:

The video tutorials are removed from the help menu.

The entire glue mesh is saved in the active zone, similar for cohesive surfaces.

Only 10 warning messages are displayed when deleting a coordinate system that is associated with a large number of points.

The following changes are made for versions 9.2.4 and higher:

The Control→Analysis Assumptions→Kinematics dialog box now contains the field "Fix Rigid Link Master Node DOF if Inactive on Attached Element". Checking this field corresponds to the command-line selection KINEMATICS RIGIDLINK-6DOF=V89, see below for a description of this selection.

In the Bolt Options dialog box, the text "Axial Bolt Stiffness Factor" is changed to "Axial Bolt Stiffness Factor for Beam-Bolt".

The following changes are made for versions 9.2.5 and higher:

The option to generate a mixture of quadrilateral and triangular elements on a body face (GFACE PREFSHAPE=QUADRILATERAL) is now supported in the Mesh Face dialog box, under the field "Element Shape: Quad and Triangular".

## **Session file writing**

The feature of writing only modified command line parameters to the session file is deactivated in version 9.2.2, as it was found that this feature did not always work correctly. So in version 9.2.2, all command line parameters are always written to the session file.

The feature of writing only modified command line parameters will be reinstated in a future release.

## **Model definition**

### **BOUNDARY-CONDITION WALL command**

In versions 9.2.2 and higher, in the BOUNDARY-CONDITION WALL command, the ALL-EXT=FREE option now automatically applies the wall boundary condition to the sliding-mesh interface for FCBI-C elements.

### **KINEMATICS RIGIDLINK-6DOF parameter**

In versions 9.2.3 and higher, the following options are available for the KINEMATICS RIGIDLINK-6DOF parameter:

NO: The FIXBOUNDARY parameter in the ADINA command is interpreted as usual. This is the default.

V89: The FIXBOUNDARY parameter in the ADINA command is always taken as V89, regardless of any user input. This option is added to provide backwards compatibility with the behavior of version 8.9.

Recall that the FIXBOUNDARY=V89 option of the ADINA command has the following effect. When the master node of a rigid link is attached to an element, the degrees of freedom on the master node are fixed if they are inactive for the attached element (e.g., x, y, z rotations are fixed if attached to a 3D solid element). Hence, when KINEMATICS RIGIDLINK-6DOF=V89 is specified, then the above effect is always included, and there is no need to specify FIXBOUNDARY=V89 in the ADINA command.

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

### **CRACK-SVS symbol**

In versions 9.2.2 and higher, the CRACK-SVS symbol is modified slightly. An extra line is added to the symbol; this line indicates the crack b direction.

# 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. Notice that report numbers are no longer used.

## **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.

The 9.2 version of the Handbook includes a chapter that discusses best practices for setting up CFD and FSI simulations that include moving meshes. This chapter reviews all aspects pertinent to moving meshes, including background mesh options, solver selection, domain subdivisions and ALE mesh constraints and conditions."

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

**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 9.2.

## **Theory and Modeling Guide**

**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**