

**A**UTOMATIC

**D**YNAMIC

**I**NCREMENTAL

**N**ONLINEAR

**A**NALYSIS

## Release Notes

ADINA 9.6.0

April 2020

ADINA R & D, Inc.

# **ADINA System 9.6**

## Release Notes

(for version 9.6.0)

April 2020

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. 2020  
April 2020 Printing  
Printed in the USA

---

# New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.6, as compared with ADINA System 9.5.4.

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

Platform	Operating system	Fortran compiler
Linux x86_64, with and without AVX extensions	2.6.32 and higher, glibc 2.12 and higher, gcc 4.4.7 and higher	Intel ifort 19.0.5.281
Windows x86_64, with and without 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 AMD64 implementations of the x86-64 architecture are supported.

2) For the Linux version, the sparse solvers are significantly faster in 9.6, for the following processor architectures:

*Intel processors, Skylake architectures and later.* In this case the sparse solvers use the AVX-512 instruction extensions. However, the speedup is significant only when the processor has 2 AVX-512 units per core.

*AMD processors that are based on the Zen architecture.*

3) For the 9.5 Linux platform, there is a considerable difference in speed between the non-AVX and AVX versions. This difference is due to the different compilers used in these two versions. For the 9.6 Linux platform, there is no significant difference in speed between the non-AVX and AVX versions, because the same compiler is used in both versions.

4) We have not released a Linux DMP version for version 9.6.

## ADINA Solids & Structures features

### Improvements for planar models

The AUI can automatically detect if a model is planar and then remove the unused master degrees of freedom. For example, if the AUI detects that the model is planar and lies in the y-z plane, then the AUI deletes the x-translation, y-rotation and z-rotation master degrees of freedom.

The above automatic detection is active when MASTER PDOF=YES and MASTER IDOF=000000.

➤ Command-line:

```
MASTER . . . PDOF
```

### Improvements for possibly unstable models

There is a new option for the preferred behavior of ADINA when a zero or negative diagonal entry in the system matrix is encountered. The new option AUTOMATIC means that ADINA chooses whether to stop or continue, depending on the model characteristics. The new option is the default.

➤ Command-line:

```
MASTER . . . IPOSIT=AUTOMATIC
```

### Improvements to line searches

Line searches are improved for problems with prescribed displacements.

### Improvements for dynamics

The  $\rho_\infty$  - Bathe method, which is a generalization of the standard Bathe method, is implemented in version 9.6 for linear and nonlinear analysis. The parameter  $\rho_\infty$  refers to the spectral radius of the method when the ratio of time step to period is very large.

The method has the following features:

When  $\rho_\infty = 0$ , the “standard Bathe method” is obtained. This method might be used primarily.

When  $\rho_\infty = 1$ , the standard Newmark Trapezoidal (TR) rule is obtained.

Roughly speaking, for values of  $\rho_\infty$  between 0 and 1, the solution characteristics of the  $\rho_\infty$  - Bathe method are in-between the standard Bathe method and the TR scheme.

For more information, see the Theory and Modeling Guide, Sections 7.3.1 and 7.4.1, and also the following references:

G. Noh and K.J. Bathe, "The Bathe Time Integration Method with Controllable Spectral Radius: the  $\rho_\infty$  - Bathe Method", *Computers & Structures*, 212, 299-310, 2019.

G. Noh and K.J. Bathe, "For Direct Time Integrations: A Comparison of the Newmark and  $\rho_\infty$  -Bathe schemes", *Computers & Structures*, 225, 2019.

➤ Command-line:

```
ANALYSIS DYNAMIC-DIRECT-INTEGRATION . . . RHO-INFINITY
```

### **Improvements to low-speed dynamics**

The backwards difference method can be used for the time integration in low-speed dynamics. The backwards difference method is first order and does not involve the accelerations. Therefore inaccuracies due to inaccurate and noisy accelerations are avoided.

When the backwards difference method is selected, the Rayleigh damping parameter ALPHA for the mass contribution to the damping matrix can be entered.

For more information, see the Theory and Modeling Guide, Section 7.2.3.

➤ Command-line:

```
AUTOMATIC TIME-STEPPING METHOD=BACKWARD ALPHA
```

### **Hydrostatic-fluid loads**

Hydrostatic fluid loading can be specified. Hydrostatic fluid loading is suitable for compressible (pneumatic) fluids in situations where the pressure and temperature of the fluid in the cavity is assumed to be uniform in the cavity.

In hydrostatic fluid loading, ADINA Structures automatically creates at all solution times the volume enclosed by the surfaces on which the hydrostatic fluid load is applied. The magnitude of the hydrostatic fluid load is then calculated using the volume and the ideal gas law.

For more information, see the Theory and Modeling Guide, Section 5.17.

- Command-line:

```
LOAD HYDROSTATIC-FLUID
```

### **Improvements to centrifugal loads**

The time variations of angular velocity and angular acceleration can be controlled separately in centrifugal loads.

- Command-line:

```
LOAD CENTRIFUGAL TFUNC-OMEGA TFUNC-ALPHA
```

### **Improvements to ANALYSIS-SWITCH**

The ANALYSIS-SWITCH feature supports switching to modal superposition analysis, and supports switching to frequency analysis with modal stress calculations.

- Command-line:

```
ANALYSIS-SWITCH ... toi=MODAL-TRANSIENT / MODAL-STRESSES
```

### **Improvements to saving of results**

The results can be saved for the solution steps equal to or later than the last solution step specified in the ELEMSAVE-STEPS and NODESAVE-STEPS commands.

- Command-line:

```
ELEMSAVE-STEPS SAVE-LAST  
NODESAVE-STEPS SAVE-LAST
```

### **Improvements for constraint equations and RBE3 elements**

The default type of constraints (basic or generalized) can be set for constraint equations and RBE3 elements.

RBE3 elements imported from a Nastran file are translated to RBE3 elements by the AUI.

- Command-line:

```
MASTER ... GENERALIZED-CONSTRAINT  
CONSTRAINT ... GENERALIZED-CONSTRAINT  
CONSTRAINT-NODE ... GENERALIZED-CONSTRAINT  
CONSTRAINT-MS ... GENERALIZED-CONSTRAINT  
RBE3 ... GENERALIZED-CONSTRAINT
```



## **Improvements for rigid links**

In version 9.6, there are two types of rigid link. The rigid link in version 9.5 and earlier is now called a “basic” rigid link. The new rigid link type in version 9.6 is called a “general” rigid link.

The differences between the two types of rigid link are now briefly explained.

In the basic rigid link, the slave node degrees of freedom are constrained using basic constraints to the master node degrees of freedom. The slave node has no independent degrees of freedom in the system matrices. Therefore a slave node cannot be fixed, or used as a master node in another constraint equation or rigid link.

In the general rigid link, the slave node degrees of freedom are constrained using general constraints to the master node degrees of freedom. The slave node has independent degrees of freedom in the system matrices. Therefore there is no restriction on connections to slave nodes. A slave node can be fixed, or used as a master node in another constraint equation or rigid link.

In the limit of an exactly converged solution, the general rigid link enforces its constraints exactly. However the stiffness matrix for the general rigid link includes compliances in the constraint equations. The advantage of using compliances is that there is no need for extra Lagrange multiplier degrees of freedom as solution unknowns. However the use of compliances has the side effect that the rate of convergence might be reduced.

In this program version, the compliances are set by the user.

The type of each rigid link can be directly specified, or the AUI can automatically determine the rigid link type. If the automatic determination is chosen, then the AUI checks, for each rigid link, whether the slave node is fixed or is used as the master node in another rigid link or constraint equation. If the slave node is fixed or used as a master node, then the rigid link is general, otherwise the rigid link is basic.

The default rigid link type can be selected with the new `RL-CONTROL` command. Note that the `RL-CONTROL` command also controls the default small/large displacement formulation used in rigid links.

For more information, see the Theory and Modeling Guide, Section 5.15.3.

### ➤ Command-line:

```
RL-CONTROL  
RIGIDLINK ... TYPE RLOPTION RLFACTOR RLTVVALUE RLRVALUE  
                RLQVALUE  
RIGIDLINK ... DISPLACEMENTS
```

```
RIGIDLINK-NODE ... TYPE RLOPTION RLFACTOR RLTVVALUE  
                    RLRVALUE RLQVALUE  
RIGIDLINK-NODE ... DISPLACEMENTS
```

### **Improvements for initial imperfections**

Shape imperfections can be both read from and written to file. In this case, the initial (stress-free) configuration is modified by the nodal displacements read from the file. The displacements of the structure are then written to a separate file for use as the initial shape imperfection of a subsequent analysis.

➤ Command-line:

```
IMPERFECTION SHAPE=READ-WRITE
```

### **Improvements for analysis zooming**

Analysis zooming can be performed from an initial shell element model to a zoomed 3D solid element model. The initial model can include both shell elements and 3D solid elements, but the zoomed model can only include 3D solid elements.

### **Improvements to mass properties calculations**

ADINA Structures prints a table in the .out file that gives the volume, mass, center of mass, and moments/products of inertia for every group and for the whole model.

The element group can be specified in the MASSINFO command, to select the element group for which the mass properties are list. Also concentrated masses can optionally selected to be included in the mass properties.

➤ Command-line:

```
MASSINFO ... EGROUP NODES
```

### **Improvements to initial conditions**

Initial conditions (initial velocities, accelerations, etc.) can be applied to rigid link slave nodes.

### **Improvements for thermal fracture mechanics with the SVS fracture option**

In the SVS method, the numerical calculation of the J-integral is affected by uniform thermal expansion. As the mesh is refined, keeping the SVS integration domains constant, the error in the numerical calculation is reduced. The numerical error can be estimated using the new 1variable SVS\_G-FE\_AV\_ETH\_ERROR.

### **Improvements to energy calculations**

The initial energies are calculated at the solution start time (initial kinetic energy from initial velocities, initial elastic strain energy from initial strains/stresses, bolt loading, etc.).

### **Improvements to membrane elements**

The following material models are now supported in membrane elements for large strain analysis:

MATERIAL PLASTIC-BILINEAR  
MATERIAL PLASTIC-MULTILINEAR  
MATERIAL THERMO-PLASTIC  
MATERIAL MULTILINEAR-PLASTIC-CREEP

### **Improvements to user-supplied material models**

Internal heat generation can be defined for user-supplied material models.

### **Improvements to Cat 2 contact**

Various improvements are made to Cat 2 contact, particularly to improve the convergence in frictional contact problems. As a result of the improvements, the results from Cat 2 contact might be different than in version 9.5.

### **Improvements to OP2 file saving**

The following additional results can be saved in OP2 files:

- Strain energies
- Stresses and strains in 3D solid elements, at the Gauss integration points
- 3D plane stress (membrane) element results
- Nonlinear modal stresses

➤ Command-line:

```
RESULTS-OP2 ... STRAINERGY
```

## **ADINA CFD & FSI features**

### **Improvements to saving of results**

The results can be saved for the solution steps equal to or later than the last solution step specified in the ELEMSAVE-STEPS and NODESAVE-STEPS commands.

➤ Command-line:

```
ELEMSAVE-STEPS SAVE-LAST  
NODESAVE-STEPS SAVE-LAST
```

## **AUI features**

### **User interface improvements**

#### **Improvements for dialog boxes**

Geometry creation and manipulation dialog boxes do not reset to their default fields after execution.

Geometry creation and manipulation dialog boxes are updated by their subordinate dialog boxes.

[P] and [V] buttons are added to various dialog boxes for point/node location coordinates and vector direction respectively.

#### **Improvements for the Model Tree**

Glue mesh sets can be renamed.

Glue mesh sets defined by element-face sets and element-edge sets can be displayed.

The model tree supports the below shortcut keys when entering text for renaming:

Ctrl C = Copy  
Ctrl V = Paste  
Ctrl X = Cut

A “Cyclic Symmetry” category is now supported in the model tree for cyclic symmetry analysis.

### **Improvements for zones**

Entities in the Model Tree can be clicked when defining a zone.

If during zone definition, the “s” keyboard button is held when clicking on an entity from the model tree, that entity is subtracted from the zone definition. For example, if the “s” keyboard button is held and the zone ENGINE is clicked from the model tree, the zone definition includes the definition:

```
SUBTRACT ZONE ENGINE
```

### **Improvements for the display toolbar**

The display tool bar has the below additional icons:

“Unmeshed Geometry” to display all geometry that does not have a mesh attached.

“Meshed Geometry” to display all the geometry that has a mesh attached.

“Face Link” to display all linked body faces.

### **Improvements for fast graphics**

When the AUI exits with fast graphics selected, it will stay in fast graphics mode in the next AUI session.

The default background color is set to black. Users can change the background color and it will be saved for future AUI sessions.

Animation can be created and saved in fast graphics mode. The model can be zoomed and rotated during animation.

Multiple mesh plots can be displayed.

The default shortcut keys for dynamic zooming, panning and rotation of model are the same as in standard graphics.

Annotations can be moved, scaled or deleted.

The configuration dialog box for fast graphics is simplified.

## **Model definition**

### **Improvements for point definitions**

Geometry points can be projected onto a line, edge, surface or face.

- Command-line:

```
POINT PROJECT
```

### **Improvement for lines definitions**

Geometry lines can be extended by following either the tangent or the curvature direction of the line.

- Command-line:

```
LINE EXTEND
```

### **Improvements for body definitions**

Additional options are available for partitioning geometry bodies. Points and edges can be removed from geometry bodies. Sheet bodies can be extended by following either the tangent or the curvature direction of the line (Parasolid bodies only).

- Command-line:

```
BODY SECTION ... OPTION POSITION OFFSET X Y Z  
BODY REMOVE-ENTITY  
BODY SHEET-EXTEND
```

### **Improvements for geometry deletion**

Points, lines, surfaces, volumes and bodies can be deleted, and the elements attached to the deleted geometry can optionally be deleted.

- Command-line:

```
GEOMDEL POINT  
GEOMDEL LINE  
GEOMDEL SURFACE  
GEOMDEL VOLUME  
GEOMDEL BODY
```

### **Improvements for skew system and axes-system definitions**

Cylindrical and spherical skew systems and axes-systems can be defined using geometry

points or nodes.

➤ Command-line:

```
SKEW-SYSTEM CYLINDRICAL-POINTS
SKEW-SYSTEM CYLINDRICAL-NODES
SKEW-SYSTEM SPHERICAL-POINTS
SKEW-SYSTEM SPHERICAL-NODES
AXES CYLINDRICAL-POINTS
AXES CYLINDRICAL-NODES
AXES SPHERICAL-POINTS
AXES SPHERICAL-NODES
```

### **Improvements for node-sets, element face-sets, element edge-sets and element sets**

When node-sets, element face-sets, element edge-sets and element sets are defined using geometry, the association between geometry and the created sets is now kept by default. Therefore if the geometry is subsequently remeshed, the sets are automatically updated.

Node-sets can be defined as the nodes close to a specified plane, for example, the x plane.

➤ Command-line:

```
NODESET ... GEOM-DEFINITION
ELFACESET .. GEOM-DEFINITION
ELEDGESET ... GEOM-DEFINITION
ELEMENTSET ... GEOM-DEFINITION
NODESET OPTION COORDINATE N1 N2 N3
```

### **Improvements for manipulations of elements**

Elements can be renumbered. The elements to be renumbered can be specified in tabular input, by element sets, by element groups or by zone.

Elements can be moved from one or more element groups into another element group.

Elements can be copied or moved using an affine transformation.

➤ Command-line:

```
ELERENUMBER
ELMOVE
ELTRANSFORM
```

### **Improvements for modifying element groups**

Multiple element groups can be modified using a single command. For example, the birth and

death times of multiple element groups can be easily modified.

➤ **Command-line:**

```
MODIFY-EGROUP (ADINA Structures)
MODIFY-EGROUP (ADINA CFD)
```

### **Improvements for applying fixities**

Fixities can be applied directly to elements, to element groups or to element sets.

➤ **Command-line:**

```
FIXBOUNDARY ELEMENT
FIXBOUNDARY EGROUP
FIXBOUNDARY ELEMENTSET
```

### **Improvements for applying loads**

There is an option for applying loads at shared boundaries when the “same load” is applied to multiple geometric entities. The loads for which this option is applicable include displacement, temperature, tgradient, pipe-internal-pressure, pore-pressure. By default, the loads are averaged.

➤ **Command-line:**

```
LOAD-OPTION SHARED-BOUNDARY
```

### **Improvements for follower forces**

If the focus point of the follower force is defined by a geometry point, and no node is attached to that geometry point, the AUI automatically creates a node at the focus point.

### **Improvements for meshing**

Some of the free-form meshing algorithms are improved in version 9.6.

One important improvement is that free-form delaunay tetrahedral meshing can now use SMP parallel processing.

If one or more bodies cannot be meshed, the AUI automatically creates a zone named `INPUT_ERROR` containing all the bodies that could not be meshed during the execution of the meshing command.

The `MESH-CONTROL` command can be used if necessary in order to generate the same meshes as in version 9.5.



➤ Command-line:

```
MESH-CONTROL
```

### **Improvements for meshing contact surfaces**

Contact surfaces can be meshed with 8-node segments when using mixed meshing. Triangular contact surfaces can be treated as degenerate quadrilateral surfaces when using mixed meshing.

➤ Command-line:

```
CSURFACE .. NODES=8 DEGENERATE
```

### **Improvements for Nastran file import**

RBAR and RBE2 elements that are translated into rigid links are now stored in rigid link-node definitions as follows. All rigid links that have the same master node number are stored in the same rigid link-node definition. In this way, all of the rigid links in a rigid link spider are stored in the same rigid link-node definition.

### **Improvements for Ansys import**

Material label number 1 is assigned to all imported element groups.

### **Improvements to data import and export**

Data can be imported and exported from dialog boxes in CSV (comma delimited) format.

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

### **Display of rigid links**

Zones can contain individual rigid links. For example, the following zone contains the rigid link with label number 2.

```
ZONE RL2  
'RIGID LINK 2'  
DATAEND
```

Ranges can be used:

```
ZONE TEST  
'RIGID LINK 20 TO 30'  
DATAEND
```

Zones can contain rigid link-sets and rigid link-nodes. A rigid-link-set is a group of rigid links created by the RIGIDLINK command, and a rigid link-node is a group of rigid links created by the RIGIDLINK-NODE command. For example

```
RIGIDLINK 3 ...
ZONE RLS3
'RIGID LINK-SET 3'
DATAEND
*
RIGIDLINK-NODE 5
ZONE RLN5
'RIGID LINK-NODE 5'
DATAEND
```

Individual rigid links, rigid link-sets and rigid link-nodes can be highlighted using the Highlight icon.

### **Improvements to the ELFORCEPOINT command**

The predefined cutting plane CUTPLANE\_ELFORCEPOINT is removed. When a cutsurface is specified in the ELFORCEPOINT command, the cutsurface must be of type cutting plane.

## **Transor for Femap features**

For Transor for Femap, the following improvements are made:

- Support for two read options for Femap Neutral file in "TRANSOR for FEMAP Post-Processing" dialog in ADINA Structures.

- Support for "Spring/Damper to Ground" element and "DOF Spring to Ground" element in ADINA Structures.

- Support for saving solution results options for NASTRAN OP2 file in "Output Requests" dialog in ADINA Structures.

- Support for element birth/death and element rupture in new "Define Property" dialog in ADINA Structures.

- Support for boundary cells, slipping boundary and leader-follower in ADINA CFD.

- Support for gap and moving wall boundary conditions in "Define Boundary Condition" dialog in ADINA CFD.

- Support for segregated solver settings for FCBI-C elements under "Solver" tab in "Analysis Settings" dialog in ADINA CFD.

Support for more options under "Flow Assumptions" tab, "FSI" tab, "Solver" tab and "Miscellaneous" tab in "Analysis Settings" dialog in ADINA CFD.

## **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**

### **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**