

AUTOMATIC

DYNAMIC

INCREMENTAL

NONLINEAR

ANALYSIS

Release Notes

ADINA 9.7.0

February 2021

ADINA R & D, Inc.

ADINA System 9.7

Release Notes

(for version 9.7.0)

February 2021

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

tel. (617) 926-5199
telefax (617) 926-0238
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. 2021
February 2021 Printing
Printed in the USA

New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.7, as compared with ADINA System 9.6.3.

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	Intel MKL library
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	MKL 2019
Windows x86_64, with and without AVX extensions	Windows 7, 8, 8.1, 10	Intel Visual Fortran 14	MKL 2020

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) The Intel MKL library used for Windows is upgraded to MKL 2020. MKL 2020 is compatible with the latest Intel and AMD processors.

3) We have not released a Linux DMP version for version 9.7.

ADINA Solids & Structures features

Improvements for automatic time-stepping

When there is no convergence when the ATS method is used without low-speed dynamics, the program can automatically switch to low-speed dynamics. The number of immediate prior converged time steps to go back to when switching to low-speed dynamics can be specified.

➤ **Command-line:**

```
AUTOMATIC TIME-STEPPING ... LSLAST-STEPS
```

Improvements for low-speed dynamics

The default mass matrix type (consistent or lumped) is changed for low-speed dynamics. In version 9.6, a lumped mass matrix is used by default for low-speed dynamics, except for models with mass-proportional loading, centrifugal loading, or rotational loading, where a consistent mass matrix is used. In version 9.7, a consistent mass matrix matrix is always used by default for low-speed dynamics.

Improvements for initial geometric imperfections

Initial geometric imperfections can be obtained using the IMPERFECTION SHAPE command in nonlinear static or implicit dynamic analysis. See the Theory and Modeling Guide, Section 11.16.

Nodal displacements can be saved at specified solution times during an analysis for use as initial geometric imperfections in a subsequent analysis.

Initial geometric imperfections can be applied to the entire model or to specified nodes in the model.

➤ **Command-line:**

```
IMPERFECTION ... TYPE ZONE READ-TIME WRITE-TIME IMP-TIMES  
IMP-TIMES
```

Improvements for substructuring

The Bathe method is supported (and is the default) for dynamic substructuring analysis.

Improvements for shell elements

User-defined material models can be used for shell elements. The DF-concrete material model can be used for single-layer shell elements.

Improvements for general rigid links

The following improvements are made for general rigid links:

By default, the general rigid link compliances are automatically calculated based on the nodal compliances of the attached structure.

Small-displacement general rigid links are supported.

Improvements for gluemeshes

General gluemeshes

In ADINA 9.7, there are two types of gluemeshes: basic gluemesh and general gluemesh. The basic gluemesh is the gluemesh supported in version 9.6, and the general gluemesh is introduced in version 9.7. The general gluemesh addresses three drawbacks of the basic gluemesh:

General gluemeshes satisfy moment equilibrium even when the gluemesh surfaces don't exactly coincide.

For general gluemeshes, there are no zero pivots from the equation solver when one of the glued components is unsupported in static analysis.

General gluemeshes can be visualized during post-processing and general gluemesh forces, gaps and tractions are available for post-processing.

However, general gluemeshes have several restrictions, including

General gluemeshes are implemented only for 3D surfaces.

General gluemeshes cause the analysis to be nonlinear even when the rest of the structure is linear.

General gluemeshes are not implemented for TMC analysis, or for cyclic symmetric analysis.

We plan to remove these restrictions in future releases.

For more information about basic and general gluemeshes, see the Theory and Modeling Guide, Section 5.16.

Mesh-detach feature

On .dat file creation, if a node on the gluemesh slave surface is also on the gluemesh master surface, then the mesh-detach feature detaches the slave node, so that the slave and master surfaces do not share nodes. This feature is on by default.

GM-CONTROL command

The GM-CONTROL command can be used to provide defaults for the corresponding parameters of the GLUEMESH command.

➤ Command-line:

```
GLUEMESH ... TYPE FORCES TRACTIONS DISPLACEMENTS ,  
SEARCHDIST SPLITSEG SPLITTOL ,  
GMCOPTION GMCFACTOR GMSVALUE GmqVALUE  
GM-CONTROL
```

Improvements for Cat 2 contact

The convergence of the Cat 2 contact algorithm when using the NSUPPRESS feature is improved.

Improvement for the RESULTS-ITERATION feature

The maximum number of equilibrium iterations saved in the porthole file for a solution step can be specified.

➤ Command-line:

```
RESULTS-ITE ... MAX-ITE
```

Improvements for the prescribed displacement force unloading feature

There are two options for force unloading at a node.

MAX-COMPONENT: Only the prescribed displacement component with maximum reaction value is replaced by an unloading force. The other prescribed displacement components are not replaced.

RESULTANT: All prescribed displacement components are replaced by unloading forces, and the direction of the unloading force is the direction of the prescribed displacement. This option is added to version 9.7 and is the default.

- Command-line:

```
APPLY-LOAD, force-direction;
```

Penetration-pressure load feature

The pressure penetration loading feature can be used to simulate the pressure of a fluid penetrating between two contact surfaces. Pressure penetration loading is useful for analysis of threaded connectors, or for analysis of soft gaskets used as a seal between stiffer structures.

Pressure-penetration loads can be applied to 2D plane stress, 2D plane strain and 2D axisymmetric elements, onto contactor contact surfaces.

- Command-line:

```
LOAD PENE-PRESSURE
```

Improvements for the hydrostatic fluid pressure loading feature

Hydrostatic fluid pressure loading can be applied to surfaces that are initially open at the beginning of the analysis, but come into contact to form a closed cavity as the analysis progresses.

Improvements for OP2 file saving

Improvements for OP2 file saving include:

The coordinate system used to save the stress/strain results of membrane (3D plane stress) elements can be specified.

For explicit analysis, results can be written to the OP2 file at each saved time step, or can be written at the end of the solution..

The beam element stresses can be written to the OP2 file, for example, for elastic-plastic beam analysis.

The truss element axial forces can be written to the OP2 file.

A new command is added to control options used for saving results in the OP2 file format.

- Command-line:

```
OP2-OPTIONS  
MASTER ... OP2SAVE
```


Improvements for NEU file saving

The truss element axial forces can be saved to the NEU file.

ADINA CFD & FSI features

Advanced moving mesh method (AMMM)

In ADINA 9.7, there are two types of moving mesh methods for CFD and fluid-structure interaction (FSI) analysis: a moving mesh method that is dependent on AUI geometry and a moving mesh method that are independent of AUI geometry.

The moving mesh method that is dependent on AUI geometry is termed OMMM (original moving-mesh method) or AUI. OMMM is the method available in version 9.6 and lower.

The moving mesh method that is independent of AUI geometry is termed AMMM (advanced moving-mesh method). AMMM is introduced in version 9.7.

AMMM allows meshes to be created using a third-party pre-processor and imported into the AUI for CFD and FSI analysis. For many simple problems, AMMM is an automatic process, so that no additional user input is required. For more complicated problems, there are new commands that control the movement of the mesh nodes.

For more information about AMMM, see the ADINA CFD Theory and Modeling Guide and the ADINA Handbook.

➤ Command-line:

```
MASTER . . . MMMETHOD
MESH-GEOMETRY
BOUNDARY-CONDITION MESH
MESH-CUT-SECTION POINTS
MESH-CUT-SECTION NODES
BOUNDARY-GEOMETRY-ORDER
MESH-LFPAIR POINTS
MESH-LFPAIR NODES
```

AUI features

Improvements for dialog boxes

In all of the “Define Mesh Density” dialog boxes, the table can be filled in without having to specify an initial entity.

Improvements for the Model Tree

A “Mesh Leader-Follower” category is now supported in the Model Tree for the advanced moving mesh feature of ADINA CFD.

Improvements for the display toolbar

The Attached Nodes option allows direct-entry of node labels. The entered nodes can be displayed (Display) or added to the current mesh plot (Display Add).

The Invert option can invert nodes, elements, and geometry. The invert action will only display the entities that have the same type as the entities in the current mesh plot.

Improvements for select mode

A zone can be created of the selected entities.

Improvements for fast graphics

The following improvements are made for fast graphics:

- Animations can be created on multiple mesh plots.

- Multiple animations are stored in an animation playlist and a particular animation can be selected for display or deletion.

- An animation playlist can be saved and loaded back for display later.

- Animations can be added to a current playlist.

Improvements to the MEASURE feature

The following additional quantities can be measured:

- Area of a surface.

- Volume of a volume.

- Shortest distance of a point to a line or edge.

- Shortest distance of a node to a line or edge.

- Shortest distance of a point to a surface or face.

- Shortest distance between two lines, two edges, or a line and an edge.

- Shortest distance between two surfaces, two faces, or a surface and a face.

- Shortest distance of a line to a surface or face.

- Shortest distance of an edge to a surface or face.

- Command-line:

```
MEASURE ... SURFACE VOLUME SURFACE2 EDGE2 FACE BODY2
```

Command line improvements

AUI script feature

With the AUI script feature, command line input can include programming syntax such as comparison operations, conditional statements, loops, echo statements and print statements. See the Command Reference Manual, Section 1.4, for details.

Model definition

New BODY SHEET-THICKEN feature

A sheet body can be thickened to create a solid body.

- Command-line:

```
BODY SHEET-THICKEN
```

Improvements for lofting bodies

A sheet body can be defined by lofting multiple lines and/or edges. A solid body can be defined by lofting multiple sheet bodies, surfaces or faces. More than two entities are allowed in the loft.

- Command-line:

```
BODY LOFTED
```

New command BODY-FILL for filling holes in bodies

A hole in a body can be filled by selecting the bounding edges of the hole.

- Command-line:

```
BODY FILL
```

Improvements for edge mesh subdivisions

The following improvements are made for edge mesh subdivision (edge mesh density):

When an edge or face of a body is split (SPLIT-EDGE and SPLIT-FACE), the edge mesh subdivisions of edges that are not split are unchanged.

When a body is transformed (BODY TRANSFORMED OPTION=MOVE), for example, if the body is translated or rotated, the edge mesh subdivisions of the body are unchanged. When bodies are defined by copying an existing body (BODY TRANSFORMED OPTION=COPY), the edge mesh subdivisions of the created bodies are taken from the parent body.

When a face is split by projecting a line or edge onto a body (BODY PROJECT), the edge mesh subdivisions of edges that are not split are unchanged.

Improvement to geometry domains used in nodal coincidence checking

Geometry domains can be defined using geometry adjacent, attached, and chained from selected geometry.

➤ Command-line:

```
DOMAIN ... OPTION ANGLE
```

New options for defining node sets

A node set can be defined as the N closest nodes to the points or nodes specified in the table input.

A node set can be defined as all nodes that lie on the plane of a local coordinate system (Cartesian, cylindrical or spherical).

➤ Command-line:

```
NODESET ... OPTION=... GROUP SYSTEM CLOSEST BOUNDARY INCLUDE
```

Improvements for rigid link definitions

The slave nodes can be selected as the closest N nodes to the master point or node.

➤ Command-line:

```
RIGIDLINK ... CLOSEST BOUNDARY
```

Improvements for RBE3 definitions

The master nodes can be selected as the closest N nodes to the slave point or node.

- Command-line:

```
RBE3 ... CLOSEST BOUNDARY
```

Improvements for GSPIDER definitions

The side 1 nodes can be selected as the closest N nodes to the side 2 point or node.

- Command-line:

```
GSPIDER ... SIDE2TYPE CLOSEST BOUNDARY
```

New FIX-MESH commands

There are additional FIX-MESH commands (commands used in ADINA-FSI to prevent nodes from moving during the analysis).

- Command-line:

```
FIX-MESH VOLUMES  
FIX-MESH BODIES  
FIX-MESH GROUP  
FIX-MESH BALLS  
FIX-MESH BOXES
```

Improvement for MESH-QUALITY feature

The mesh quality of elements in a zone can be checked.

- Command-line:

```
MESH-QUALITY ... MESH-ZONE
```

Improvements for the MESH-CONVERT feature

The element order can be changed from first-order to second-order, or vice versa.

- Command-line:

```
MESH-CONVERT ... ORDER
```

Improvements for Nastran file import

Nastran PIDs are translated to the DESCRIPTION parameter of element groups, material models, node sets, element sets, element-edge sets, and element-face sets.

Model display and post-processing

Display of element edges

The element edge color can be different than the element face color.

➤ Command-line:

```
ELDEPICTION ... ELEDORCOLOR ELEDECOLOR
```

Band table display

The number of significant digits in a band table can be specified.

➤ Command-line:

```
BANDTABLE ... SIGDIG
```

Display of gluemeshes

Zones can contain gluemeshes, gluemesh surfaces or gluemesh segments. For example,

```
ZONE GM1  
'GLUEMESH 1'  
DATAEND
```

```
ZONE SURFACE1  
'GLUEMESH SURFACE 1 OF GLUEMESH 1'  
DATAEND
```

```
ZONE SEGMENT1  
'GLUEMESH SEGMENT 1 OF GLUEMESH SURFACE 1 OF GLUEMESH 1'  
DATAEND
```

Surface 1 is the slave surface and surface 2 is the master surface.

Similarly, gluemeshes, gluemesh surfaces and gluemesh segments can be highlighted using the Highlight icon.

Display of prescribed temperatures and prescribed heat fluxes

Band plots of prescribed temperatures and prescribed heat fluxes can be displayed.

Improvements for Re-Frame icon

The Re-Frame icon only translates the displayed entities to the center of the screen and re-zooms. The Re-Frame icon does not change the rotation or shading

Display and Select toolbars

The Display toolbar and Select toolbar can be used in post-processing.

Preferences options

Display preferences can be changed in post-processing using the Specify Preferences dialog box.

Graphics Windows Settings

The width and height of the mesh plot can be specified using the Graphics Windows Settings dialog box.

Transor for Femap features

The following improvements are made:

TRANSOR for FEMAP 9.7.0 supports FEMAP 2020.2 and ADINA 9.7.0.

Added support of the concrete material model for 2D and 3D solid elements.

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 Femap Users Guide