ADINA

A UTOMATIC

D YNAMIC

I NCREMENTAL

N ONLINEAR

A NALYSIS

Release Notes

ADINA 9.8.2   June 2022

ADINA R & D, Inc.

ADINA System 9.8

Release Notes

(for version 9.8.2)

June 2022

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

tel. (617) 926-5199
telefax (617) 926-0238
www.adina.com

New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.8, as compared with ADINA System 9.7.3.

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, 11	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.

ADINA Solids & Structures features

Joule heating

The Joule heating feature in ADINA Structures heat transfer has been substantially improved in version 9.8.

The electrical potentials and current densities are now stored on the porthole file, along with the temperatures and heat fluxes.

Theory and Modeling Guide for ADINA Structures heat transfer / TMC

Chapter 12 of the ADINA Structures TMG has been rewritten in order to provide a more complete and accurate description of the ADINA Structures heat transfer / TMC feature.

Improvements for gluemeshes

General gluemeshes

Some of the restrictions for the general gluemeshes in version 9.7 have been removed in version 9.8. In version 9.8,

> General gluemeshes are implemented for 2D and 3D surfaces.

> General gluemeshes are implemented for heat transfer and TMC analysis, and for cyclic symmetric analysis.

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

Gluemesh birth-death

The reference configuration for gluemesh birth can be chosen. In version 9.7 and earlier, the reference configuration is the original configuration. In version 9.8, the reference configuration is, by default, the configuration at the time of gluemesh birth.

➢ Command-line:

```
GM-CONTROL … BIRTHCONFIG
GLUEMESH … BIRTHCONFIG
```

Improvement for 3D bolt element

The placement of the bolt split plane in 3D bolt element groups is changed slightly for certain models. The bolt split plane is shifted closer to the center of the bolt, along the bolt direction. This improvement may cause the results from these models to change slightly.

Improvements in concrete material models

Rupture can now be specified for the concrete and DF-concrete material models. Rupture occurs when the material is crushed.

➢ Command-line:

```
MATERIAL CONCRETE … RUPTURE
MATERIAL DF-CONCRETE … RUPTURE
```

Improvement for the $\rho - \infty$ Bathe method

New parameter ETA can be used instead of parameter RHOINF to specify constants for the $\rho - \infty$ Bathe method. The $\rho - \infty$ Bathe method with parameter ETA is useful in wave propagation analysis.

➢ Command-line:

```
ANALYSIS DYNAMIC-DIRECT-INTEGRATION ETA
```

Improvements for the hydrostatic fluid pressure loading feature

The accuracy of hydrostatic fluid pressure results is improved for 2D axisymmetric analysis. In 9.7, only the enclosed 2D area is taken into account when calculating the current hydrostatic pressure (see the ADINA Structures Theory and Modeling Guide Eq. 5.17-2a). In 9.8, the whole volume of the enclosed 3D domain (a rotation around the axisymmetric axis) is taken into account, which is proportional to the distance from the centroid of the 2D area to the rotating axis (see the ADINA Structures Theory and Modeling Guide Eq. 5.17-2b).

Improvements for the results-iteration feature

The equilibrium iteration results can be saved for selected solution times. It is allowed to specify the increment of equilibrium iterations (for example, when increment=3, then the results for the 1st, 4th, 7th equilibrium iterations are saved).

➢ Command-line:

```
RESULTS-ITE … OPTION=TABLE-TIME
```

Improvements for OP2 file saving

Improvements for OP2 file saving include:

The solution times at which element results and nodal results are saved to the OP2 file can be specified.

The stress/strain results for 4-node shell elements can be output either at the center of the element or at the corner nodes also.

When results are saved both to the OP2 file and to the porthole file, the results can be saved at different solution times into the two files.

➢ Command-line:

```
OP2-OPTIONS … ELEMOP2SAVE-STEPS NODEOP2SAVE-STEPS
               CQUAD4-STR
ELEMOP2SAVE-STEPS
NODEOP2SAVE-STEPS
```

AUI features

User interface improvements

New toolbar icons

The Display toolbar has a new Element Types icon . This icon can be used to select certain element types for display. For example, the user can choose to display only tetrahedral elements or to display both tetrahedral and wedge elements.

The General toolbar has a new Rotate XYZ icon . This icon can be used to rotate the model about the global X, Y, or Z axis by a specified angle (default 10 degrees) with each click of the icon. When the Shift key is pressed, the direction of rotation is reversed. The axis, direction and angle of rotation can be changed through the Rotation Settings icon .

Improvements for dialog boxes

The Define Zone dialog box has a "Automatically Include All Elements Attached to Nodes" option that includes all the elements attached to the nodes specified in the Zone definition.

The Define Node Set, Define Element Set, Define Element Face Set, and Define Element Edge Set dialog boxes have a Display and Display Add option to display or add to the display the current set defined.

The List Zone Values dialog box allows column-select of lists output through the "Export Table..." feature.

Improvements for the display toolbar

When the Display toolbar actions are executed (Adjacent, Attached, Auto-Chain), etc., the AUI add the executed action to the mesh plot (Display Add). The AUI will not replace the mesh plot with the executed action (Display).

In the Attached Nodes dialog box of the Display Toolbar, a Create Zone option is added to create a zone of all entities displayed after the Attached Nodes dialog box is executed.

Improvements for select mode

The right-click Create Zone option is added for all entities in Select mode.

Improvements for fast graphics

Improvements made for fast graphics include

> Added support for the new icon Rotate XYZ
> Added ability to move annotations in the graphics window using the middle mouse button
> Added support for Mesh Zoom, Refit and Unzoom Partial icons
> Accelerated animation generation using multithreading
> Improved animation visualizations for big models

Improvements to the MEASURE feature

The total length of line segments defined by successive nodes or geometry points can be determined.

➤ Command-line:

```
MEASURE … GTYPE={MULTI-POINTS/MULTI-NODES}
point_i
```

Command line improvements

AUI script feature

Advanced substitutions are allowed within do loops. This allows the user to substitute a variable name from the value of another variable. The substitution begins and ends with a % sign:

Example:

```
auiscript end
$Bolt_Thr_Pro_1_z1 = 11
$Bolt_Thr_Pro_1_z2 = 12
$Bolt_Thr_Pro_1_z3 = 13
$Bolt_Thr_Pro_1_z4 = 14
$Bolt_Thr_Pro_2_z1 = 11
$Bolt_Thr_Pro_2_z2 = 12
$Bolt_Thr_Pro_2_z3 = 13
$Bolt_Thr_Pro_2_z4 = 14
do $ii = 1,2
    do $jj = 1,4
        $Boltx= $Bolt_Thr_Pro_%ii%_z%jj%
    enddo
enddo
auiscript end
```

Model definition

Import of Nastran models

The BCPROP entry can be used to define a contact or glue surface on membrane (3D plane stress) elements in addition to shell elements.

The BSURF entry can be used to define a contact or glue surface on membrane (3D plane stress) elements in addition to shell elements.

The source and target regions referenced in a BGSET (gluemesh set) entry can be defined by the BCPROP or BSURF entry, in addition to the BCPROPS or BSURFS entry.

Additional way to define a geometry point

A geometry point can be defined using the POINT COORDINATE command. When the POINT COORDINATE command is used to define a geometry point, the corresponding entry in the session file is much shorter than when the COORDINATE POINT command is used to define a geometry point, particularly when there are already many geometry points defined.

➢ Command-line:

```
POINT COORDINATE
```

New body transformation feature

The faces in a body can be moved. (This feature is available for ADINA-M for Parasolid only.)

➢ Command-line:

```
FACE-TRANSFORMED
```

New sheet body features

A sheet body can be modified by specifying points where radius blends are to be created. Faces can be removed from a sheet body. (These features are available for ADINA-M for Parasolid only.)

➢ Command-line:

```
BODY PBLEND
BODY SHEET-TRIM
```

New option for defining element face-sets

The element faces corresponding to those displayed in the current mesh plot can be chosen as an element face-set.

➢ Command-line:

```
ELFACESET … EXTERNAL
```

Improvements for the GBODY command

The free-form mesher used in the GBODY command is improved in version 9.8. By default the meshes produced by GBODY are different than in version 9.7, however, the 9.7 algorithms can be specified in order to obtain the 9.7 meshes.

➢ Command-line:

```
MESH-CONTROL GBODY-VERSION=V97
```

New meshing command for shell and membrane elements

Shell and 3D plane stress (membrane) elements can be generated from element face-sets.

➢ Command-line:

```
GELFACESET
```

Model display and post-processing

Display of mass-proportional loads

Mass-proportional loads are now plotted by the LOADPLOT command.

Display of boundary conditions

The boundary depiction dialog box now allows for the selected display of boundary condition codes according to the description in the AUI Command Reference Manual for Display Processing, BOUNDEPICTION command.

Transor for Femap features

The following improvements are made:

TRANSOR for FEMAP 9.8.0 supports FEMAP 2022.1 and ADINA 9.8.0.

Added support for rigid end zones of the ADINA beam element. For detailed information, see chapter 1.2.7.6.3 "Define Rigid End Zones for Beam Element" in the TRANSOR for Femap Users Guide.

Replaced "9/27-Node Element Conversion" in the "Model Settings" menu with a new "Element Conversion" dialog box. For detailed information, see "Element Conversion" in chapter 1.2.1 "Model Settings Menu" in the TRANSOR for Femap Users Guide.

Added support of failure criteria (maximum stress, maximum strain, Tsai-Hill, Tsai-Wu and Tsai-Hashin) for the FEMAP laminate element. For detailed information, see chapter 1.2.7.4 "ADINA Composite Shell Element" in the TRANSOR for Femap Users Guide.

Improvements between 9.8.0 and 9.8.1

Improvements in the contact penetration pressure feature

The contact penetration pressure feature can now be used in the following case: 2D contact surface forming a closed loop.

Improvements in op2 files

PVT data block

The op2 file now contains the PVT (Parameter Variable Table) data block, and supports unit conversion when the parameter UNITSYS is defined in the Nastran input file.

op2 file results when a Nastran file is imported

For a model created without the import of a Nastran file, and when op2 results are requested (MASTER RESULTS=OP2 or OP2+PORT), the default settings for results saved in the op2 file follow the default settings for the porthole file.

However, when a Nastran file is imported (using the NASTRAN-ADINA command), the default settings for results saved in the op2 file are overwritten by the settings in the imported Nastran file. In particular, if no results are requested in the Nastran file, then if op2 results are requested, then the AUI will give an error during ADINA data file generation.

Transor for Femap features

The following improvements are made:

TRANSOR for FEMAP 9.8.1 supports FEMAP 2022.1 and ADINA 9.8.1.

Renamed the menu "FSI Boundary Conditions" under "Model Parameters" to "Fluid-Structure Boundary.

Added support of ADINA cast iron material for 2D solid and 3D solid elements. For detailed information, please see chapter 1.2.5 Materials Menu.

Added support of Noh-Bathe method for dynamic explicit analysis.

Added support of time integration method for low speed dynamics.

Added "Analyze" button in "ADINA Analyze" dialog box. It will automatically execute the functions of "Create .in file", "Create .dat file" and "Run model" buttons. The corresponding OP2 file will be automatically imported if it is specified in the "Output Requests" menu.

Improvements between 9.8.1 and 9.8.2

Improvements in concentrated masses and concentrated dampers for ADINA Structures

The limitation of 99999 concentrated masses / concentrated dampers is removed.

Improvement for automatically redrawing graphics

AUI 9.8.2 redraws the graphics automatically more often, without the need to click the Redraw icon. This improvement affects the Primer instructions, however, the Primer is not yet updated to correspond to the redraw improvement.

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 https://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
Primer problem 69 "Joule heat analysis of a radio-frequency ablation experiment" is added.

Theory and Modeling Guide (TMG)
    Volume I: ADINA Solids & Structures
    Volume II: ADINA Thermal
    Volume III: ADINA CFD&FSI
    Volume IV: ADINA EM
Updates are made to describe the new features of the solution programs.

ADINA Verification Manual

ADINA-Nastran Interface Manual

TRANSOR for Femap Users Guide