ADINA

AUTOMATIC

DYNAMIC

INCREMENTAL

NONLINEAR

ANALYSIS

Release Notes

ADINA 9.3.2   July 2017

ADINA R & D, Inc.

ADINA System 9.3

Release Notes

(for version 9.3.2)

July 2017

New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.3, as compared with ADINA System 9.2.5.

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.

ADINA Solids & Structures features

Improvements to dynamic analysis

The global mass and damping matrices can be scaled as functions of time. This feature can be useful for problems in which the inertia forces and damping forces should only be applied at specific times during the analysis, and also for problems in which the inertia forces and damping forces should be smoothly transitioned during the analysis.

➤ Command-line:
      ANALYSIS DYNAMIC-DIRECT-INTEGRATION ... INERTIA-TFUNC
                                      DAMP-TFUNC

Improvements to nonlinear modal superposition analysis

The full Newton method can be used. This method is most useful when the following features are used: beam end releases, u/p elements, incompatible mode elements.

Improvements to the CMS method

Either a full Bathe subspace iteration analysis or a Rayleigh-Ritz analysis can be used to calculate the lowest eigenvalues and eigenvectors of the original finite element system in the second phase of the CMS method.

An option to only compute the reduced stiffness and mass matrices in the CMS method is added. With this option, no eigenvalues and eigenvectors of the original finite element system are computed.

See the ADINA Structures TMG, Section 11.18, for more information.

By default, the reduced system matrices are output.

➤ Command-line:
      FREQUENCIES ... CMS-SUBSPACE NEIGEN

Improvements to solution diagonstics

Detailed solution diagnostics are switched on by default.

➤ Command-line:
      DIAGNOSTICS ... SOLUTION CONTACT

Improvements to the output (.out) and message (.msg) files

Additional information is written to the output and message files, including

> ► Warning messages in the .out file when the maximum displacement is larger than 5% of the model size and when small displacement kinematic assumptions are used.
> ► Messages when the restart file is created or updated.
> ► Messages when the porthole or op2 file is created or updated.

The format of the printout of the overdistorted elements in the .out file is improved, so that the information can be directly cut-and-pasted into a zone, for example:

```
*** List of distorted elements           ***
*** Please cut and paste into a zone in AUI ***
ELEMENT        1 OF ELEMENT GROUP      1
ELEMENT        2 OF ELEMENT GROUP      1
ELEMENT        5 OF ELEMENT GROUP      1
```

Improvements for automatic time-stepping

In 9.3 and higher, the ATS method is used by default in nonlinear static and implicit dynamic analysis.

There is a new option NSUBSTEP, used in conjunction with existing option RESTORE=NO, that controls for how many time steps the reduced (subdivided) time step size is used after an ATS cutback. If NSUBSTEP>0, the reduced time step size is used for NSUBSTEP consecutive substeps. See the ADINA Structures TMG, Section 7.2.4 for further details.

> ➢ Command-line:
>   MASTER ... AUTOMATIC
>   AUTOMATIC TIME-STEPPING ... NSUBSTEP

Improvements for analysis zooming

Temperatures are mapped in thermo-mechanical coupling analysis (ADINA TMC).

The ATS method can be used for analysis zooming

Improvements for restart analysis

The restart file solution times (solution times for which the restart file contains information, and therefore solution times from which restart analyses can begin) can be directly specified.

The RESTART-INFO command in the AUI can be used to list information in a specified restart file, such as the data the restart file was written, the problem heading, the number of nodes, etc.

➢ Command-line:
    MASTER ... IRTIMES
    IRTIMES
    RESTART-INFO

Improvements to equilibrium iterations

The minimum number of equilibrium iterations to use during each time step (or during each sub-step if the Bathe method is used) can be specified. The default is to use at least one equilibrium iteration (as in 9.2).

When beam end releases are used to release the moment degrees of freedom in large displacement beam elements, the program will use at least two equilibrium iterations per time step (or sub-step). This prevents possible premature convergence.

➢ Command-line:
    ITERATION ... MIN-ITERATIONS

Skyline solver

The skyline solver (MASTER SOLVER=SKYLINE) is obsolete. If the skyline solver is selected, the program automatically uses the sparse solver.

Improvements to energy calculations

The following improvements are made to energy calculations:

► Header at the top of the .egy file giving information about the model.
► Energy calculations for low-speed dynamic analysis include the kinetic and damping energies.
► Energy calculations are added for beam elements.
► Energy calculations are added for shell elements (except for multilayer shells, 8-node shells, 3D-shells and transition shells).
► Energy calculations are added for 2D contact
► Energy calculations are added for the three-network model, and for the rubber-like models that include viscous effects.
► The energy calculations can be toggled on/off in a restart analysis.

Improvement to element birth-death

The program now adds a tolerance to the user-input birth and death times. In this way the user-input birth and death times can coincide with solution times. For example, if the user-input birth time equals a solution time, then the element is guaranteed to be active at that solution time, and if the user-input death time equals a solution time, the element is guaranteed to be inactive at that solution time.

The description of element birth-death has been improved and clarified, see the ADINA Structures TMG, Section 11.4.

Connector elements

The connector element is a new element group type in version 9.3. This element is, in essence, a large displacement spring and damper element.

The connector element has two subtypes: joint and matrix. In the joint-type connector element, the stiffness and damping coefficients are either constant or multilinear. In the matrix-type connector element, the stiffness and damping matrices are directly specified.

See the ADINA Structures TMG, Section 2.14, for more details.

➢ Command-line:
   EGROUP CONNECTOR
   CONN-DEFINE
   CONN-PROP JOINT
   CONN-PROP MATRIX
   JOINT-NL-C
   JOINT-NL-K)

Radiosity elements

The radiosity-type radiation elements previously available in ADINA Thermal are now available in ADINA Structures for ADINA TMC analysis. 2-D and 3-D radiation elements are implemented.

➢ Command-line:
   RGROUP RADIATION2
   RGROUP RADIATION3
   RADIATIONSURFACE
   RADIATION-NODES
   RSDELETE
   RSURFACE

Improvements to three-network model

A new option is added, that specifies whether the material parameters are calibrated in accordance with the three-network model paper by Bergström and Bischoff, or whether the material parameters are calibrated using the MCalibration software developed by Veryst Engineering. The default is to use the convention of the three-network model paper by Bergström and Bischoff (same as in version 9.2).-

➢ Command-line:
   MATERIAL THREE-NETWORK ... CALIBRATION

Improvements to the creep-variable user-supplied subroutines

The creep-variable user-supplied subroutines now are passed the solution time, so the the creep parameters A0 to A7 can be functions of the solution time.

Improvements to gluemesh definitions

As a user convenience, gluemeshes can be defined using contact groups and contact surfaces. That is, a contact surface pair can be easily converted into a gluemesh pair, using a special parameter in the contact group definitions. Note that mesh glueing is frequently more accurate than using tied contact.

Iso-beam elements (plane strain, plane stress and axisymmetric) can be glued.

➢ Command-line:
     CGROUP CONTACT2 ... GLUEMESH
     CGROUP CONTACT3 ... GLUEMESH

Improvements to frequency analysis with contact

In frequency analysis, contact compliance can either be considered or neglected (the default is to consider contact compliance, as in version 9.2).

ADINA automatically switches off contact damping for frequency analysis and modal particiaption factor analysis.

➢ Command-line:
     FREQUENCIES CONT-COMPL

Improvements to contact

Contact traction improvements

For symmetric contact and self-contact, the contact tractions are drastically improved in 9.3. The contact tractions at a contactor node now include the effects of contact forces resulting from the contact interaction in which that node is a target node. This improvement is made for 2D and 3D contact.

These improvements apply when the default contact surface options are used: new contact surfaces, current contact traction calculations.

Slip velocity improvements

In 9.3, the slip velocity of a contactor node is defined as the velocity of the contactor node relative to the target surface. In 9.2, the slip velocity is defined oppositely, namely, as the velocity of the target surface relative to the contactor node.

This improvement does not change the solution results; this improvement is made because it is more natural to define the slip velocity using the definition implemented in 9.3.

Improvements to load definitions with the arrival time feature

In version 9.2, if an arrival time is specified in a load definition, the arrival time is always used. In version 9.3, there is an option to ignore arrival times specified in the load definitions. The default is to use the arrival times (as in version 9.2).

➢ Command-line:
     LOAD-OPTION ... TFSHIFT

Improvements to bolts

The bolt damping feature is improved, as described in the ADINA Structures TMG, Section 11.17.3.

The following is a specific example. In version 9.3, when low-speed dynamics is selected without bolt damping, the program uses the damping from the low-speed dynamics feature during the bolt steps (in 9.2, the program uses no damping during the bolt steps in this situation). Therefore convergence during the bolt steps is easier in version 9.3.

Additional checking is done for the meshing of 3D-bolt elements, so that if 3D-bolt elements use invalid meshing, the .dat file is not generated.

➢ Command-line:
     BOLT-OPTIONS ... DAMPING

Improvements to reactions

The inertia and damping forces acting on the prescribed or fixed degrees of freedom are now always included in the reaction forces.

Improvements to op2 file writing

In 9.3, the solution results in the Nastran OP2 file can be controlled using the new RESULTS-OP2 command.

Modal stresses are now saved in the OP2 file.

➢ Command-line:
     RESULTS-OP2

ADINA CFD & FSI features

Conditions, diffusion loads and flow resistance loads

In 9.3, two types of loads are added.

> ► Flow resistance loads, which add a frictional force to the right-hand side of the momentum equations
> ► Diffusion loads, to add viscosity or thermal conductivity to the flow.

These loads which are active when certain conditions are satisfied. The following conditions are supported.

> ► The distance between two boundaries
> ► The outer-iteration process

Flow resistance loading or diffusion loading with a boundary distance condition can be used to model flow stoppage due to solid-to-solid contact, flow leakage that might occur due to imperfect solid-to-solid contact in the physical parts, etc. Diffusion loading with an outer iteration condition can be used to improve the convergence of the solution for problems that diverge in the first few outer iterations.

When diffision loading is used with the boundary distance condition in turbulence models, "laminar-like" flow is assumed when the boundary distance condition is closed. This is known as conditional turbulence. Conditional turbulence can improve the convergence of the solution for problems with thin gaps, where the flow is expected to be physically laminar in the gap.

> ➢ Command-line:
> > LOAD DIFFUSION
> > LOAD FLOW-RESISTANCE
> > CONDITION BOUNDARY-DIST
> > CONDITION OUTER-ITERATION
> > CONDITION COMBINED

Improvements for outer-iterations

In 9.3, a skewness scheme for improving convergence is available. The skewness scheme can improve convergence of the outer-iterations if the mesh is highly distorted, however the cost per outer-iteration is increased. The default is not to use the skewness scheme.

> ➢ Command-line:
> > OUTER-ITERATION ... SKEWNESS-CORRECTION

Realizable k-ε turbulence model

The realizable k-ε turbulence model is implemented. For more information on this model, see the ADINA CFD & FSI TMG, Section 5.6.10.

➤ Command-line:
  MATERIAL TURBULENT-KE ... TYPE

Data input checks for initial conditions

The following data input checks are implemented:

 ► Data input checks on the initial temperature and pressure are added for compressible flow.
 ► Data input checks on the initial turbulence are added for turbulent flow.
 ► Data input checks are added for load and boundary conditions.

Improvements to the sliding-mesh interface

The fluid solution accuracy and convergence rate is improved when the sliding-mesh interface is curved, and when there is an abrupt change of element size across the sliding-mesh interface

Improvements to moving meshes

The efficiency of the solution of moving meshes is improved for problems in which the boundary surface is very large.

Improvements to adaptive meshing using SAM

In 9.3, the solution times at which the adaptive meshing will be performed can be specified.

➤ Command-line:
  ADP-CONTROL ... ADP-MESH-TIMES
  ADP-MESH-TIMES

Improvements to temperature interpolation in the structural model for TFSI analysis.

In 9.2 and lower, temperatures at the mid-side nodes of higher-order elements are interpolated from the corner nodes, regardless of the settting of TMC-CONTROL TEMP-INTER. In 9.3, temperatures at the mid-side nodes of higher-order elements are interpolated from all element nodes, if TMC-CONTROL TEMP-INTER=NO.

➤ Command-line:
  TMC-CONTROL ... TEMP-INTER

Solution monitor for CFD and FSI

A solution monitor for CFD and FSI is available, to monitor the equation residuals, variable residuals and total mass conservation residuals of the solution. This monitor is implemented for the FCBI-C elements.

Improvements to porthole file writing

In 9.3, the nodal and element results written to the porthole file can be controlled using the new RESULTS-NODE and RESULTS-ELEMENT commands.

➢ Command-line:
      RESULTS-NODE
      RESULTS-ELEMENT)

Improvements to output of reaction forces and FSI forces

In version 9.2 and lower, the reaction forces output by the structural model in FSI analysis includes the FSI forces from the fluid model.

In version 9.3 and higher, the reaction forces output by the structural model do not include the FSI forces and the FSI forces are output separately. In this way, the reaction forces and FSI forces can be post-processed separately.

AUI features

User interface improvements

Improvements to the Model Tree

Improvements to multi-zone

When ZZ_MULTI_ZONE is displayed, the zones that were originally selected to define ZZ_MULTI_ZONE are reselected, and the user can dynamically modify this selection (e.g. add or remove zones).

When ZZ_MULTI_ZONE is selected with other zones, the other zones are appended to ZZ_MULTI_ZONE (in 9.2, the other zones overwrite ZZ_MULTI_ZONE).

ZZ_MULTI_ZONE is automatically updated when the zones that define ZZ_MULTI_ZONE are updated.

Improvements to zone groups

Zone groups can be renamed. Zones contained in zone groups can be modified, renamed, deleted, etc. Zone groups allow "Copy to" and "Move to" operations, in which "Copy to" copies the zone to another zone group while keeping the zone in the original zone group, and "Move to" moves a zone to another zone group.

Improvements to zones

When a zone is renamed, the color on the zone is the same as the color of the original zone. After the color of a zone is changed in the Model Tree, the zone is displayed unhighlighted.

The zone colors for each zone are now displayed in all program versions (in 9.2, the zone colors are only displayed in the Linux version).

Improvements for picking

Improvements for picking elements

An additional right-click option is available: `Invert`. This option can be used to invert the current selection.

Improvements for picking element face-sets

The following additional right-click options are available: `Adjacent`, `Attached`, `Face angle [20]`, `Invert`. The Face angle option has an input box that can be used to adjust the angle.

Improvements for picking element edge-sets

The following additional right-click options are available: `Adjacent`, `Attached`, `Auto-Chain`, `Face angle [20]`, `Invert`.

Improvements for picking geometry

The following additional right-click options are available: `Adjacent`, `Attached`, `Auto-Chain`, `Face angle [20]`, `Invert`.

Message displaying total number of entities picked

When picking entities, the Message Window displays the total number of entities picked.

Improvements to the dialog boxes

The following dialog boxes no longer reset to default settings after execution:

Nodes→Split Mesh
Nodes→Detach Mesh
Nodes→Join Mesh

ADINA-M→Boolean Operator→Operator Type: Merge
ADINA-M→Boolean Operator→Operator Type: Intersect
ADINA-M→Boolean Operator→Operator Type: Subtract

ADINA-M→Define Body→Type: Sheet, Defined by: Body Faces

Meshing→Elements→Element Set→Method: Adjacent
Meshing→Elements→Element Face Set→Method: Auto-Chain Element Faces

The following dialog boxes are updated by their subordinate dialog boxes:

Define Line→Type: Transformed→Transformation Label
Define Surface→Type: Transformed→Transformation Label
Define Volume→Type: Transformed→Transformation Label
Apply Fixity→Define…
Apply Initial Condition→Define…
Define Result Control→Coord Sys. for Transformed Results
Define Model Point (Mesh Extreme Point)→Zone Name
Define Model Point (Reaction Sum Point)→Zone Name

The vector selection button [V] is now added to the following dialog boxes:

Define Line→Type: Revolved
Body Sweep→Sweep Faces Along: Vector
Define Axes System (Orthonormal Set of Vectors) →Define By: Vectors
Define Axes System (Orthonormal Set of Vectors) →Define By: Cylindrical
Define Axes System (Orthonormal Set of Vectors) →Define By: Spherical
Define Coordinate System→Vector A, Vector B
Mesh Lines→Orientation Vector
Mesh Edges→Orientation Vector

Improvements to the skew system and axes-system dialog boxes

The Apply Skew Coordinate System, Apply Material Axes System and Apply Initial Strain Axes System dialog boxes now have the same form as the Apply Load, Apply Fixity and Apply Initial Condition dialog boxes. The usage of these dialog boxes is: first the system is defined, then the geometric entities associated with that system are specified.

The Apply Skew Coordinate System dialog box now only shows the skew system numbers actually defined by the user.

Improvements for deleting geometry

When deleting geometry, a pop-up dialog box allows for the elements attached to geometry to be optionally deleted.

Improvements for deleting nodes and points

When deleting geometry points, a pop-up dialog box allows for the nodes attached to a geometry point to be optionally deleted.

Improvements for deleting elements

All commands that delete elements (whether from geometry or from element lists) are now accessed by the Delete Mesh dialog box.

Improvements to the equilibrium iteration graph

A new equilibrium iteration window allows for the selection of the number of equilibrium iterations to view.

Improvements to Fast Graphics

For Linux, the FGM shortcut keys can be customized.

Improvements to the solver dialog boxes

When floating licenses are used, the program immediately releases the solver floating license key after the job has completed, even if the solver dialog box is still open. (In 9.2, the solver license key is only released after the solver dialog box is closed.)

Model definition

Improvements for import of Nastran files

In 9.2, by default, elements from different bulk entry (e.g CHEXA, CPENTA, CTETRA) but having the same PID are split into different element groups, for ADINA Structures and ADINA Thermal models. In 9.3, the default is changed so that the elements are not split into different element groups.

➢ Command-line:
    NASTRAN-ADINA ... SPLIT

Improvements for import of Parasolid files

When importing geometry from a Parasolid file, the number assigned to the first created point can be specified (default = highest existing point number + 1).

On Windows, long file names are now always used (in 9.2, the file names are in the DOS 8.3 format).

➢ Command-line:
    LOADSOLID ... PLABEL

Improvements for coordinate system definition

A local coordinate system can be defined using three nodes. This option is similar to the existing option in which a local coordinate system can be defined using three geometry points.

➢ Command-line:
    SYSTEM ... MODE={ ... / 4 }

Improvements for skew system definitions and assignments

New commands SKEW-SYSTEM NODES, SKEW-SYSTEM POINTS, SKEW-SYSTEM NORMAL, SKEW-SYSTEM VECTOR, SKEW-SYSTEM EULERANGLE SKEW-SYSTEM SPHERICAL, SKEW-SYSTEM CYLINDRICAL can be used to define skew systems using various options. These new commands correspond to the new skew system dialog boxes described above.

New commands DOF-SYSTEM TWO-D and DOF-SYSTEM THREE-D can be used to assign skew systems to geometry lines, edges, surfaces and faces. These new commands correspond to the new axes-system dialog boxes described above.

➢ Command-line:
    DOF-SYSTEM TWO-D
    DOF-SYSTEM THREE-D
    SKEW-SYSTEM NODES
    SKEW-SYSTEM POINTS)
    SKEW-SYSTEM NORMAL
    SKEW-SYSTEM VECTOR
    SKEW-SYSTEM EULERANGLE
    SKEW-SYSTEM SPHERICAL
    SKEW-SYSTEM CYLINDRICAL

Improvements for axes-system definitions and assignments

Axes-systems can be defined in terms of Euler angles.

New commands SET-AXES-MATERIAL THREE-D, SET-AXES-STRAIN THREE-D can be used to assign axes-systems to geometry surface and faces. These new commands correspond to the new axes-system dialog boxes described above.

➢ Command-line:
      AXES EULERANGLES
      SET-AXES-MATERIAL THREE-D
      SET-AXES-STRAIN THREE-D

Improvements to BODY LOFTED

The *reverse$_i$* data column is removed from the BODY LOFTED command. The AUI automatically determines the reverse flag for each lofted entity.

Improvements to BODY SWEEP and BODY REVOLVED

The distance in which the geometry face (used to define the body) is swept can now be specified inthe BODY SWEEP command.

For both BODY SWEEP and BODY REVOLVED, an option is added in which existing bodies that are identical to the newly created body are deleted. (An existing body is only deleted if it does not have an associated mesh.) This option is the default.

➢ Command-line:
      BODY SWEEP ... DISTANCE DELETE-DUPLICATE
      BODY REVOLVED ... DELETE-DUPLICATE

Improvements to BODY SECTION

Multiple bodies can be selected when using BODY SECTION.

➢ Command-line:
      BODY SECTION ... DOMAIN

Improvements to BODY TRANSFORMED

When a body is moved, the mesh associated with the body is also moved. When a body is transformed by an affine transformation (scale, reflection, direct, points), the mesh is likewise transformed.

Improvements for geometry deletion

All loads, boundary conditions, glue surfaces, contact surfaces, etc. are automatically updated when geometry is deleted, so that the deleted geometry is no longer referenced by the model.

Improvements for definitions of rigid links

In 9.3, there are additional types of entity used to specify slave nodes and master nodes. The additional types are: specified nodes, element edge-sets and element face-sets.

A new parameter LINK-OPT is added to the RIGIDLINK command, to provide control over how multiple nodes on the slave and master entities are connected. The LINK-OPT parameter provides the same choices as the existing OPTION parameter, but the parameter values names are more intuitive (e.g. LINK-OPT=PARAMETRIC is the same as OPTION=0).

In order that the ADINA FIXBOUNDARY=V89 option be used by default, KINEMATICS RIGIDLINK-6DOF can be set to V89. The option KINEMATICS RIGIDLINK-6DOF=V89 is not the default, but this option is provided for user models created using previous versions of ADINA.

➢ Command-line:
  RIGIDLINK ... SLAVETYPE MASTERTYPE
  RIGIDLINK ... LINK-OPT
  KINEMATICS ... RIGIDLINK-6DOF

Definitions of truss and beam elements using spiders

An element spider is set of elements that connects the nodes on two sides of a model. The new command GSPIDER creates spiders of truss or beam elements, in which each side can be a point, a surface, a face, a volume, a body or a node set.

➢ Command-line:
  GSPIDER

Improvements to truss and spring element definitions

Additional options are available when defining trusses, springs and 6-DOF springs from geometry lines and node-sets. These options control the creation of duplicate elements and coincident nodes.

➢ Command-line:
  TRUSS LINE ... DUPLICATE NCOINCIDE NCTOLERANCE
  SPRING LINE ... DUPLICATE NCOINCIDE NCTOLERANCE
  SPRING NODESET ... DUPLICATE
  SPRING-6DOF LINES ... DUPLICATE NCOINCIDE NCTOLERANCE
  SPRING-6DOF NODESET ... DUPLICATE

Improvements for node set, element edge-set, element face-set, element set definitions

In version 9.3, there are additional options for node set, element edge-set, element face-set and element set definitions, as follows.

▶ Node sets can now be defined using geometry points, nodes in geometry volumes or nodes in geometry bodies.
▶ Element edge-sets can now be defined using element groups, nodes, adjacent element edges, attached element edges, chained element edges or geometry points.
▶ Element face-sets can now be defined using nodes, adjacent element faces, attached element faces, geometry points, geometry volumes or geometry bodies. Also, when using auto-chaining, the external element faces selected are based either on element groups or on the whole model (the default is the whole model).
▶ Element sets can now be defined using geometry points, element face-sets or element edge-sets.

When elements are deleted from the model, the definitions of element edge-sets, element face-sets and element sets are updated so that the sets do not reference the deleted elements.

➢ Command-line:
  NODESET ... OPTION={.../ POINT / VOLUME / BODY
  ELEDGESET ... OPTION={... / GROUP / NODES / ATTACHED / ADJACENT /
              CHAIN / POINTS}
  ELFACESET ... OPTION={... / NODES / ATTACHED / ADJACENT /
              POINTS / VOLUME / BODY}
  ELFACESET ... EXTERNAL
  ELEMENTSET ... OPTION={... / POINT / ELFACESET / ELEDGESET}

Improvement for wall boundary condition definitions

For the FCBI-C elements, the wall boundary condition is applied to sliding-mesh surfaces when ALL-EXT=FREE in the wall boundary condition definition.

Improvements for rebar definitions

Additional options for nodal coincidence checking are available when generating rebars on geometry lines. These options allow rebar elements to be connected to other elements in the model.

➢ Command-line:
  REBAR-LINE ... NCTOLERANCE GROUP

Improvements for face-links

Additional options control the creation of face-links, to improve the ease-of-use.

➢ Command-line:
FACELINK ... OPTION={ONE-FACE / ONE-BODY / TWO-FACES /
FACE-BODY / TWO-BODIES / ALL}

Improvement for generation of nodes using GPOINT

In 9.2, when GPOINT is used with NCOINCIDE=ALL, and if a node is already present, the command terminates with an error message. In 9.3, this behavior is changed, so that the AUI uses the existing node with highest number.

Meshing of swept / extruded / lofted geomery

New command GLOFTED can mesh (with either prismatic or hexahedral elements) any body that can be seen as the result of a modeling operation that sweeps / extrudes / lofts a "source" (body) face. For example, the following bodies can be meshed:

► A body that is topologically equivalent to a cube or prism.
► A body that results from the BODY SWEEP command.
► A body that results from the BODY REVOLVED command.
► A body that results from the BODY LOFTED command.

➢ Command-line:
GLOFTED

Improvments for meshing rigid contact surfaces

Additional warning messages are added, when meshing rigid contact surfaces onto geometry with underlying elements or contact segments.

Improvements for pressure, temperature, heatflux loads

For ADINA Structures, pressure, temperature and heat flux loads can be mapped from a point cloud stored in a file onto the model. The point cloud data can be obtained in any convenient way, for example, from an analytical function, from experimental results or from numerical results computed using ADINA or other simulation software.

➢ Command-line:
LOAD PRESSURE ... MAPPING
LOAD TEMPERATURE ... MAPPING
LOAD HEATFLUX ... MAPPING
LOAD-MAPPING

Improvement for renumbering of nodes

In 9.3, the NODE-RENUMBER command can be used to renumber all nodes, including nodes attached to geometry (in 9.2, the NODE-RENUMBER command can only be used to renumber nodes not attached to geometry).

➢ Command-line:
    NODE-RENUMBER

Checking for duplicate elements during data file generation

Version 9.3 can check for duplicate elements in the entire model , or in an element group, during data file generation. Using this feature, if duplicate elements are found, data file generation is stopped. The default is to check for duplicate elements in the entire model.

➢ Command-line:
    CONTROL ... DUPLICATE

Model display and post-processing

Improvements to variable categories

The following variables are moved into a new "Contact" category:

NODAL_CONTACT_GAP
NODAL_CONTACT_STATUS
NODAL_CONTACT_OVERLAP
SEGMENT_LENGTH
SEGMENT_AREA
NORMAL-{XYZ}
CONTACT_STATE_FLAG

The following variables are moved into a new "Energy" category:

STRAIN_ENERGY_DENSITY
MAX_STRAIN_ENERGY_DENSITY
DEV_STRAIN_ENERGY_DENSITY
MAX_DEV_STRAIN_ENERGY_DENSITY
FREE_ENERGY_DENSITY
ELASTIC_FREE_ENERGY_DENSITY

Variable PLASTIC_WORK is moved into the "Dissipation/Generation" category.

Post-processing of FSI forces

FSI forces from the structural model in FSI analysis can be plotted as vectors using the REACTIONPLOT command.

The FSI force components can be listed or graphed using the variables FSI_FORCE-{XYZ}, FSI_FORCE-{ABC} and the magnitude of the FSI force can be listed or graphed using variable FSI_FORCE_MAGNITUDE.

➢ Command-line:
REACTIONPLOT ... QUANTITY={ ... / FSI-FORCE}

Plotting of concentrated masses and dampers

Concentrated masses and dampers are now plotted by default.

Improvement for highlighting

Element sets and FS boundaries can now be highlighted.

Improvement to DUMPLIST command

The starting line of the file dumped using the DUMPLIST command can now be specified. Lines before the starting line are not dumped.

➢ Command-line:
DUMPLIST ... STARTLINE

Improvements between 9.3.0 and 9.3.1

ADINA Solids & Structures features

Improvements to restart feature

ADINA Structures 9.3.1 checks to make sure that the restart file is generated by 9.3.0 or later. If the restart file is generated by an earlier version of ADINA Structures, the program stops with an error message.

Improvements between 9.3.1 and 9.3.2

ADINA DMP

Improvements to installation and documentation of ADINA DMP

► The *ADINA DMP System 9.3 Installation Notes* have been substantially revised and expanded. The document *Distributed Memory Processing in ADINA DMP 9.3* is renamed to *ADINA DMP System 9.3 User Guide*, and this document has also been substantially revised and expanded.

► The layout of the third-party libraries distributed with ADINA DMP 9.3 is revised. In particular, the entire Open MPI 1.4.3 distribution (as built at ADINA R&D) is now included with ADINA DMP 9.3.

► The `setup_dmp` script is rewritten. You can now choose to use either the Open MPI 1.4.3 distributed with ADINA DMP, or the Open MPI 1.4.3 at your site. We recommend that you use Open MPI 1.4.3 compiled and installed at your site, because Open MPI 1.4.3 installed at your site is customized for the networking at your site. `setup_dmp` also gives you the options to select or deselect any of the BTL or MTL components used by Open MPI.

► The `addmp9.3` script now works properly when the Open MPI MTL psm component is used for message passing (important for InfiniBand networking using InfiniPath).

► The `addmp9.3` script includes a new option that binds the process threads to specific CPU cores. Binding can sometimes improve performance. Binding is not used by default.

► Utililty programs `ompi_info_wrapper` and `openmpiprobe9.3` are now included in the distribution of ADINA DMP. These utility programs can aid in troubleshooting ADINA DMP, in particular the launching of ADINA DMP with Open MPI. For details, see the *ADINA DMP System 9.3 User Guide*.

Improvements to the ADINA DMP solvers

► The user-supplied material model feature can now be used with ADINA DMP.

► In ADINA-CFD and ADINA-FSI, the moving mesh sparse solver is DMP parallelized.

ADINA Solids & Structures features

Improvements to thermo-plastic material models with kinematic hardening

For thermo-plastic material models with kinematic hardening, a back stress temperature correction can now be included. Without the back stress temperature correction, in an analysis in which the temperatures are changing in time, the results can be non-physical. For details, and for an example, see the ADINA Structures Theory and Modeling Guide, Section 3.6.2. By default,. the back stress temperature correction is turned off, but the back stress temperature correction can be turned on using the command KINEMATICS BACKSTRESS=YES.

➢ Command-line:
> KINEMATICS ... BACKSTRESS

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.

The 9.3 version of the Handbook includes new sections on body face meshing, lofted meshing and mid-side node placement.

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 9.3. New primer problem 66 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
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