

**A**UTOMATIC

**D**YNAMIC

**I**NCREMENTAL

**N**ONLINEAR

**A**NALYSIS

## Release Notes

ADINA 9.5.0

April 2019

ADINA R & D, Inc.

# ADINA System 9.5

## Release Notes

(for version 9.5.0)

April 2019

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. 2019  
April 2019 Printing  
Printed in the USA

---

# New and updated feature summary

This section lists the new and updated features that are available in ADINA System 9.5, as compared with ADINA System 9.4.4.

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, SMP 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 7, 8, 8.1, 10	Intel Visual Fortran 14
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.

### Table of supported platforms, DMP program version

Platform	Operating system	Fortran compiler
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

- 1) The DMP program version is based on OpenMPI version 3.0.0.
- 2) The DMP program version does not support all features of the SMP program versions, see the DMP Users Guide for details.
- 3) The DMP program version requires the AVX extensions.

## ADINA Solids & Structures features

### Change of default for frequency analysis

The enriched subspace method implemented in version 9.4 is now the default in frequency analysis. The enriched subspace method is as robust as the standard subspace method, but the solution time required is often reduced by a factor of 3 or more for large models, as compared to the standard subspace method.

➤ Command-line:

```
FREQUENCIES ... METHOD=ENRICHED-SUBSPACE
```

### Improvements for analysis zooming

The following improvements are made for analysis zooming:

11-node tetrahedral elements are supported.

Time interpolation of the initial model results are supported, so that ATS analysis of the zoomed model can be performed.

Analysis zooming for ADINA TMC heat transfer only analysis is supported.

### Improvements to periodic symmetry

The following improvements are made to periodic symmetry:

The 3D-iterative solver is supported.

Large pivots for nodes on cyclic symmetry boundary lines are no longer listed in the stiffness matrix maximum diagonal printout in the .out file.

### **Improvements for reaction force calculations**

The inertia and damping forces on prescribed degrees of freedom are included in the reaction calculation for explicit dynamic analysis.

### **Improvements for the analysis switch feature**

The following improvements are made in the analysis switch feature:

A frequency analysis can be performed at any time during the solution, and frequency analyses can be performed at multiple times during the solution.

The eigenvectors of the previous frequency analysis can be used as starting vectors.

A modal stress analysis can be performed at any time during the solution, or after the final converged solution. Modal stress analyses can be performed at multiple times during the solution.

Modal participation factor analysis can be performed.

➤ **Command-line:**

```
ANALYSIS-SWITCH  START-VECTOR  
timei toi  
ANALYSIS MODAL-PARTICIPATION ... NSTEP DTSIZE)
```

### **Improvements for stiffness stabilization**

It is allowed to use stiffness stabilization for nonlinear analysis only.

➤ **Command-line:**

```
MASTER STABILIZE=NONLINEAR
```

### **Improvements to ADINA TMC**

The following improvements are made to ADINA TMC using ADINA Structures:

The temperatures are saved to the .tem file for use in a subsequent structural-only analysis.

It is allowed to restart from a structural-only analysis to ADINA TMC, and it is allowed to restart from ADINA TMC to a structural-only analysis.

### **Improvements for solution accuracy indicators**

The node number and direction of the maximum value for each solution accuracy indicator is printed.

### **Efficiency improvements for highly coupled degrees of freedom**

When a degree of freedom is coupled to many other degrees of freedom, for example, though a rigid link spider or an element spider, the efficiency of the solution is improved. In 9.4 the solution time is dependent on the node numbering, in 9.5, the solution time is not dependent on the node numbering.

### **Improvements for bolts**

The results can be saved for bolt-steps even when bolt tables are not used.

➤ Command-line:

```
BOLT-OPTIONS SAVE
```

### **Improvements for general elements**

The data format for the matrix input is increased from 10 digits to 20 digits per matrix entry, thus allowing greater precision in the input.

### **New thermo-plastic cast iron material model**

A new thermo-plastic cast iron material model is available. The thermo-plastic cast iron material model can be used for 2D plane strain/axisymmetric elements and for 3D elements.

The thermo-plastic cast iron material model is fully described in Section 3.18 of the ADINA Structures Theory and Modeling Guide.

➤ Command-line:

```
MATERIAL CAST-IRON  
MATERIAL THERMO-CAST-IRON
```

### **Plastic strain and creep strain magnitude**

The plastic strain and creep strain magnitudes are now saved to the porthole file, for all elements / material models for which there are plastic strains and creep strains.

## **Improvements for contact**

Starting from version 9.5, there are two contact versions, labeled “Cat” contact versions (Cat stands for Contactor and Target).

*Cat 1 contact:* Cat 1 contact represents ADINA developments through program version 9.4.

*Cat 2 contact:* Cat 2 contact represents the ADINA developments starting from program version 9.5.

Many modeling features are available in both Cat 1 and Cat 2 contact. However some modeling features are available only in Cat 1 contact and other modeling features are available only in Cat 2 contact.

Our long-term goal is to make all best practice modeling features available in Cat 2 contact. However, in the short term, there are some useful modeling features that are available only in Cat 1 contact, for example double-sided contact.

The following table gives the major improvements in Cat 2 contact, compared to Cat 1 contact.

<b>Feature</b>	<b>Improvement in Cat 2 contact</b>
Constraint function method	Lagrange multipliers are tractions, not forces  Improved scaling / preconditioning of stiffness matrix.
Continuous contact normals	Option to use median normals instead of averaged normals in 3D contact. The median normals minimize the difference between the continuous normals and attached segment normals, and therefore median normals are preferred for problems in which there are sharp corners in the contact surfaces.
Higher-order contact segments	Improved equivalent areas for higher-order contact segments
Contact search	Contact search based on field of view, more robust than nearest-node contact search.  Contact search is SMP parallelized within contact groups.



Feature	Improvement in Cat 2 contact
Normal contact (pertaining to the normal contact response)	<p>New backstop option</p> <p>Improved constraint functions</p> <p>Normal contact properties, such as compliance, associated with contact pairs</p> <p>New command CNORMAL to specify the normal contact parameters.</p> <p>The compliance factor CFACTOR1 can be multiplied by a time function in order to change CFACTOR1 during the analysis.</p> <p>In explicit contact, CFACTOR1 can be used to specify the compliance.</p>
Friction	<p>Improved frictional constraint functions</p> <p>Nonsymmetric frictional coupling terms included when nonsymmetric solver is chosen</p> <p>Improved update of contact coordinate system for friction</p> <p>New commands CFRICTION to specify the frictional contact parameters.</p> <p>In explicit contact, EPST can be used to specify the frictional sticking velocity.</p>
Relative contact damping	<p>New feature. New command CRELDAMP BASIC added to support the new feature.</p>
Absolute contact damping	<p>Option for traction-based contact damping</p> <p>Time functions for contact damping.</p> <p>New command CABSDAMP BASIC to specify the absolute contact damping parameters.</p>
Explicit contact	<p>Penalty method is the only explicit contact algorithm.</p> <p>Continuous normals in explicit contact.</p> <p>Contact slip loads in explicit contact.</p> <p>The tangential penalty is now a damping factor.</p>

<b>Feature</b>	<b>Improvement in Cat 2 contact</b>
Small displacement contact	Small displacement contact in restart can either rebuild the contact constraints or keep the existing contact constraints.  Small displacement contact can be used with FSI direct coupling
Initial penetrations feature	TIME-PENETRATION=FIRST-STEP so that the penetration time is the first user-specified step. If ATS cutbacks occur within the first user-specified step, then the initial penetration is applied gradually within the first user-specified step.
Contact slip loads	Contact slip loads include the effects of offsets.
2D contact	The thickness of the contact surfaces can be entered for plane stress contact.

The contact chapter (Chapter 4) in the Theory and Modeling Guide has been completely rewritten. Section 4.1 in that chapter contains a complete list of features, indicating which features are supported in Cat 1 contact and which features are supported in Cat 2 contact.

The CGROUP and CONTACT-CONTROL command descriptions in the Command Reference Manuals have been completely reorganized.

➤ **Command-line:**

```

CONTACT-CONTROL ...
    CAT-VERSION CNCALC MAXCNANGLE
    CNORMAL CFRICTION CRELDAMP CABSDAMP
    PATCH-RELEASE
    SEARCHDIST OPPNTEST
CGROUP CONTACT2 ...
    THICKNESS NODAL-AREA CNCALC MAXCNANGLE
    CNORMAL CFRICTION CRELDAMP
    PATCH-RELEASE
    SEARCHDIST OPPNTEST
CGROUP CONTACT3 ...
    NODAL-AREA CNCALC MAXCNANGLE
    CNORMAL CFRICTION CRELDAMP
    PATCH-RELEASE
    SEARCHDIST OPPNTEST
CONTACTPAIR ...
    CNORMAL CFRICTION CRELDAMP
    CNORMAL BASIC
    CFRICTION COULOMB
    CFRICTION LAW1
    CFRICTION LAW2
    CFRICTION USER-SUPPLIED
    CRELDAMP BASIC
    CABSDAMP BASIC
    
```

## **Improvements for initial stresses**

The following improvements are made:

Initial stresses at the integration points can be applied at the integration points of shell elements, including multilayer shell elements and 3D-shell elements.

Initial stresses can be applied for the following isothermal plastic material models: plastic-bilinear, plastic-multilinear, plastic-orthotropic and plastic-cyclic.

➤ Command-line:

```
INITIAL CONDITION
tinti layeri
```

## **Joule-heat analysis using the TMC feature of ADINA Structures**

Joule-heat analysis can be performed using the TMC feature of ADINA Structures

➤ Command-line:

```
LOAD CURRENT-DENSITY
LOAD ELECTRIC-POTENTIAL
LOAD NODAL-CURRENT
APPLY-LOAD CURRENT-DENSITY
APPLY-LOAD ELECTRIC-POTENTIAL
APPLY-LOAD NODAL-CURRENT
```

## **Improvement for convection and radiation loads**

The environmental temperatures for convection / radiation loads can either be added or averaged if there is more than one load application at a node. The default is to average, as in version 9.4.

➤ Command-line:

```
LOAD-OPTION CONVECTION RADIATION)
```

## **Improvements to concentrated force loads and other load types**

The data format for concentrated loads is increased to 20 digits. Previously concentrated force loads were written using 10 digits, meaning that force loads entered more than 10 digits were rounded (example: 1.2345678901234E10 was rounded to .123457E11). The same improvement is made for other load types.

## Solution graph feature

ADINA 9.5 can create solution graphs of structural and TMC analysis results for static and implicit dynamic analysis. Solution graphs can be created for displacements, velocities, accelerations, temperatures, reactions, etc. Solution graphs can be viewed during runtime and after the solution is complete using the ADINA Graph Viewer.

For more information about the solution graph feature, see Section 7.7 of the TMG.

- Command-line:

```
GRAPH  
LOCATION
```

## Improvement for saving nodal and element results

It is allowed to specify the solution times, or solution steps, for which nodal and element results are output.

- Command-line:

```
ELEMSAVE-STEPS OPTION={STEPBLOCK / TIMEBLOCK}  
NODESAVE-STEPS  OPTION={STEPBLOCK / TIMEBLOCK}
```

## Option for saving equilibrium iteration results to the porthole file

ADINA 9.5 can save the results for each equilibrium iteration when the model does not converge, or for a specified solution time. Viewing the equilibrium iteration results can give insight as to why a model does not converge.

For more information about this feature, see Section 7.2.1 of the TMG.

- Command-line:

```
RESULTS-ITE
```

## Improvements for EnSight output

The following improvements are made for saving EnSight files:

It is now possible to save only the EnSight file, without saving the porthole file.

Additional structural results, such as contact tangential tractions, maximum shear stresses and FSI forces can be saved.

The results to be saved in the EnSight file can be selected using the RESULTS-ENSIGHT command.

➤ Command-line:

```
RESULTS-ENSIGHT ... DISPLACEMENT VELOCITY ACCELERATION  
STRESS REACTION CONTACT TEMPERATURE  
BOLT FSI
```

### **Improvements in TRANSOR for Femap**

TRANSOR for Femap has a number of improvements, including:

Support for RBE3 elements.

Support for the Include File functionality, allowing the user to include AUI commands in the .in file written by Femap.

Support for porthole output, neutral file and porthole output, or op2 file and porthole output, from ADINA Structures.

### **Improvements for op2 files**

The following improvements are made:

The model geometry (nodal coordinates, element connectivity, etc.) are written to the GEOM1 and GEOM2 data blocks in the op2 file, so that Fe-Safe can read the model geometry.

The ADINA connector element is translated into a CBUSH element, and the element forces/moments of the connector element are saved in the op2 file.

## **ADINA CFD & FSI features**

### **FLOW-RATE boundary condition**

The mass flow rate or volume flow rate can be prescribed at a boundary. This feature is useful for compressible flow problems, and for problems in which the boundary area changes during the solution.

➤ Command-line:

```
BOUNDARY-CONDITION FLOW-RATE
```

## **Improvement to wall boundary condition**

The slipping boundary mesh constraint is supported for wall, moving wall and FSI boundaries, when the FCBI-C element is used.

➤ Command-line:

```
BOUNDARY-CONDITION WALL ... SLIP-BOUND  
BOUNDARY-CONDITION FLUID-STRUCTURE ... SLIP-BOUND  
SLIP-BOUND
```

## **Improvements to the VOF method**

The following improvements are made to the VOF method:

MCURVE materials for the different VOF-species are supported. This allows the VOF-species to be functions of temperature and pressure.

The VOF species solution variables from a prior analysis can be mapped as initial conditions onto a new model.

## **Total energy equation for Reynolds element viscous heating**

The viscous heating term using the real velocity profile can now be used for the heating source term in the total energy equation, the nodal reaction force and element stress, etc.

## **Improvements to the out file**

All messages printed by ADINA CFD now use the label numbers input by the user for element groups, material curves, etc.

## **Improvement for saving nodal and element results**

It is allowed to specify the solution times, or solution steps, for which nodal and element results are output.

➤ Command-line:

```
ELEMSAVE-STEPS OPTION={STEPBLOCK / TIMEBLOCK}  
NODESAVE-STEPS OPTION={STEPBLOCK / TIMEBLOCK}
```

## **Improvements for EnSight output**

The following improvements are made for saving EnSight files:

Additional results, such as stresses, reaction force, effective viscosity, can be saved.

Reaction forces are saved at wall/FSI boundaries, mass fluxes for FCBI-C elements are saved at external boundaries.

Additional geometry entities, such as the element face sets in which the velocity, pressure, temperature boundary conditions are applied, can be saved.

### **Improvements to the solution graph feature**









The mass flow rate and volume flow rate for individual VOF-species can be graphed.

## **AUI features**


### **User interface improvements**

#### **New icons**

The following new icons are added in a new Picking toolbar:

- Adjacent  **ADJ**
- Attached  **ATT**
- Attached to Nodes  **ATT\***
- Auto-chain 
- Auto-chaining angle  **ANG**
- Invert  **INV**
- Same EGroup 
- Same Material 

When in the highlight (selection) mode of a dialog box (that is, after double-clicking in a cyan table), the above icons can be used instead of (or in addition to) the right-click options to change the highlighted entities.

When using the Pick icon , the Display/Picking toolbar icons can be used to control the display. The options have the following display actions:

**Adjacent:** Appends to the meshplot all elements that are adjacent to the elements currently displayed.


**Attached:** Appends to the meshplot all elements that are attached to the elements currently displayed.

**Attached to nodes:** Appends to the meshplot all elements that are attached to the specified nodes.


**Invert:** Displays the “inverse” of the current display. For example, if the zone BOLTS is currently displayed, clicking this icon displays everything except for zone BOLTS.

**Same EGroup:** Appends to the meshplot all elements in the same element group as the elements currently displayed.

**Same Material:** Appends to the meshplot all elements with the same material as the elements currently displayed.

When using the Select icon , the Display/Picking toolbar icons can be used to change the highlighted entities.

#### *Re-frame icon*

The Re-frame icon  reframes the currently displayed entities so that they are displayed in the center of the graphics window, unzoomed, just like when the entities were first created.

#### *Color Zones icon*

The Color Element Groups icon is renamed to Color Zones . The Color Zones icon assigns a color to every zone.

### **Improvements to the Model Tree**

*Display Add option:* Appends the selected entity to the current meshplot.

*Display option for element face-sets, element edge-sets and node-sets.* Displays the selected face-set, edge-set or node-set.

*Display option for mesh glue:* Displays the mesh glue sets

### **Improvements to zones and zone groups**

The following right-click picking options can now be used when defining a zone:

- Adjacent
- Attached
- Auto-chain
- Invert
- Same Element Group



Same Material  
Attached to Nodes  
Edge Angle / Face Angle, ... (depends on the entity type)

A zone from the Model Tree can be picked to include that zone in the current zone definition. The 's' key can be used to subtract a Model Tree zone from the current zone definition.

### **Improvements for picking**

The Select mode icon  supports the Display and Hide right-click options.

There are additional right-click picking options

Same Element Group  
Same Material  
Attached to Nodes

### **Custom navigation tool leys**

The navigation tool keys (pan, resize, rotate(XY), rotate(Z)) can be customized. For example, the following settings can be made so that the AUI functions the same as Ansa:

Pan = Ctrl + Middle-mouse button  
Rotate = Ctrl + Left-mouse button

### **Auto-completion in dialog boxes.**

The following dialog boxes now include auto-completion. With auto-completion, the user does not need to remember the exact syntax of the strings used for specification.

Define Zone  
Define Resultant

### **Improvements for the Windows version**

A large icon button option is available to display large icons for the toolbars.

## **Model definition**

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

Parameter TINTEG of the NXSTRAT command now specifies the time integration method of all analyses with inertia/damping effects (full dynamics, low-speed dynamics and the TLA-S method).

### **Options for moving and copying geometry points**

Geometry points can be moved or copied using a geometric transformation.

- Command-line:

```
POINT TRANSFORMED
```

### **Improvements to BODY SWEEP and BODY-ROTATE**

The faces of 11-node tetrahedral elements can be swept using BODY SWEEP and BODY ROTATE.

### **Improvement to BODY PROJECT**

Lines / edges can be projected onto the face of a body.

- Command-line:

```
BODY PROJECT
```

### **Improvements for SPLIT-EDGE**

The following improvements are made to the SPLIT-EDGE command.

Multiple edges can be split by a single command.

An edge can be split by a nearby geometry point.

- Command-line:

```
SPLIT-EDGE ... OPTION={PARAMETRIC/POINT} USPLIT POINT  
DELETE-POINT  
edgei bodyi
```

### **Improvements for SPLIT-FACE**

The following improvements are made to the SPLIT-FACE command.

The program can automatically detect the face to be split.

A face can be split by a nearby geometry edge.

- Command-line:

```
SPLIT-FACE ... DETECT OPTION EDGE
```

### **Additional options for creating nodes**

Nodes can be defined between existing nodes or geometry points. A node can be defined at the center of a circle that passes through three nodes or three geometry points.

- Command-line:

```
NODE BETWEEN  
NODE CENTER
```

### **Improvements for rigid link definitions**

Rigid links can be defined using the nodes within a search distance of master nodes / master points.

- Command-line:

```
RIGIDLINK ... SLAVETYPE=NEARBY DISTANCE
```

### **Improvements for RBE3 element definitions**

RBE3 elements can be defined using the nodes within a search distance of slave nodes / slave points.

- Command-line:

```
RBE3 ... MASTERTYPE=NEARBY DISTANCE BOUNDARY GROUP
```

### **Improvements for spider definitions**

Spiders can be defined using the nodes within a search distance of master nodes / master points.

- Command-line:

```
GSPIDER ... SIDE2TYPE=NEARBY DISTANCE BOUNDARY SIDE2GROUP
```

### **Improvements for face-link definitions**

Face links can be created by specifying multiple bodies. All faces of the specified bodies are considered for face linking. A face link is created between any two geometrically and topologically identical faces in the considered list.

➤ **Command-line:**

```
FACELINK ... OPTION={.../BODIES}  
bodyi
```

**Improvements for element face-set definitions**

When defining face sets, auto-chaining can be restricted to include elements of the specified element group.

➤ **Command-line:**

```
ELFACESET ... EXTERNAL
```

**Improvements for element set definitions**

An element set can be defined using a search radius DISTANCE from the nodes or geometry points specified in the input.

➤ **Command-line:**

```
ELEMENTSET ... DISTANCE
```

**Improvements to MESH-CONVERT**

When converting 8-node element faces to 9-node element faces, the newly created nodes are by default added to the existing node sets.

➤ **Command-line:**

```
MESH-CONVERT SKEW=YES LOAD-INIT=YES NODESET
```

**Improvements to mesh gluing contact surfaces**

Mesh gluing contact surfaces are supported for contact surfaces defined using element-face nodes.

**Mesh quality checking**

The AUI can create bar graphs of mesh quality metrics, and save elements that fail mesh quality criteria to a zone or element set.

For more information, see Section 2.8.4 of the ADINA Handbook.

- Command-line:

MESH-QUALITY

### **Remeshing feature**

The GELEM command can be used to remesh elements. This command can be used to remesh elements of poor quality identified by the MESH-QUALITY command, or to locally refine the mesh.

For more information, see Section 2.8.4 of the ADINA Handbook.

- Command-line:

GELEM

### **Compatible mesh checking for ADINA-CFD fluid meshes**

On .dat file generation, the AUI can check the fluid mesh for compatibility. If the fluid mesh is not compatible, .dat file generation stops with an error message. The default is to check the fluid mesh if the number of elements is less than 1,000,000.

For more information, see Section 2.8.1 of the ADINA Handbook.

- Command-line:

CONTROL . . . COMPATIBLE-MESH

### **Solution Graph Viewer**

The following improvements are made to the Solution Graph Viewer:

The Solution Graph Viewer can plot solution graphs produced by the Solution Graph feature.

The Solution Graph Viewer can plot bar graphs produced by the Mesh Quality feature.

The Solution Graph Viewer can plot the equilibrium iteration results from FSI analysis.

More options are added for customizing the axes, legends and graph colors.

The Solution Graph Viewer can be accessed from the post-processor (Graph→Solution Graph View).

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

### **BOLTINFO command**

The BOLTINFO command outputs a table of the bolt number and element group number. The BOLTINFO command can be used in pre- and post-processing.

➤ Command-line:

```
BOLTINFO
```

## **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**

Problem instructions are revised for the ADINA System 9.5.

### **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**