




FEMAP

Release Notes

Version 7.1



FEMAP Version 7.1 Release Notes

Copyright © 1986-2000 by Structural Dynamics Research Corp.

Proprietary Data. Unauthorized use, distribution, or duplication is prohibited.

All Rights Reserved.

Portions of this software and related documentation are derived from GHS3D software under license from INRIA, other portions are copyrighted by and are the property of Electronic Data Systems Corporation and Spatial Technology Inc.

The FEMAP Documentation may not be copied, reproduced, disclosed, transferred, or reduced to any form, including electronic medium or machine-readable form, or transmitted or publicly performed by any means, electronic or otherwise, unless Structural Dynamics Research Corp (SDRC) consents in writing in advance.

Use of the software has been provided under a Software License Agreement.

Information described in this document is furnished for information only, is subject to change without notice, and should not be construed as a commitment by SDRC. SDRC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

FEMAP is a registered trademark of Structural Dynamics Research Corp.

Dual Engine Geometry Modeling and *FEMAP Professional* are trademarks of Structural Dynamics Research Corp.

Structural Dynamics Research Corp.

P.O. Box 1172, Exton, PA 19341

Phone: (610) 458-3660

FAX: (610) 458-3665

E-mail: info@femap.com

Web: <http://www.femap.com>

This manual and software product are both copyrighted and all rights are reserved by SDRC. The distribution and sale of this product are intended for the use of the original purchaser only and for use only on the computer system specified. The software product may be used only under the provisions of the license agreement included with the FEMAP package. Unless otherwise stated, you may only use this software on a single computer, by one person, at one time.

Trademark Information

Throughout this manual, and the software, you will see references to other applications and trademarks which are the property of various companies.

- NASTRAN and Cosmic NASTRAN are registered trademarks of NASA.
- IBM is a registered trademark of International Business Machines Corporation.
- MSC/, MSC/NASTRAN, MSC/pal, MSC/pal 2, and PATRAN are registered trademarks of The MacNeal-Schwendler Corporation.
- CDA/Sprint is a trademark of The CDA Group.
- UAI/NASTRAN is a product of Universal Analytics, Inc.
- CSA/NASTRAN is a product of Computerized Structural Research and Analysis Corp.
- ME/NASTRAN is a product of Macro Engineering, Inc.
- ABAQUS is a registered trademark of Hibbitt, Karlsson, and Sorenson, Inc.
- CAEFEM is a trademark of Concurrent Analysis Corp.
- SDRC, SDRC I-DEAS and I-DEAS are registered trademarks of Structural Dynamics Research Corporation.
- SSS/NASTRAN is a trademark of Schaeffer Software Systems, Inc.
- ANSYS is a registered trademark of ANSYS, Inc.
- STAAD and STARDYNE are products and trademarks of Research Engineers, Inc.
- COSMOS and COSMOS/M are registered trademarks of Structural Research and Analysis Corporation.
- WECAN is a registered trademark of Westinghouse, Inc., marketed by AEGIS Software Corp.
- ALGOR is a registered trademark of Algor Interactive Systems, Inc.
- CFDesign is a trademark of Blue Ridge Numerics, Inc.
- Pro/ENGINEER is a registered trademark of Parametric Technology Corporation
- GENESIS is a registered trademark of Vanderplaats, Miura and Associates, Inc.
- MTAB*Stress is a trademark of Structural Analysis, Inc.
- AutoCAD and DXF are registered trademarks of Autodesk, Inc.
- Solid Edge is a trademark of Intergraph Corporation.
- MicroStation is a registered trademark of Bentley Systems, Inc.
- SolidWorks is a trademark of SolidWorks Corporation.
- MARC is a trademark of MARC Analysis Research Corporation.
- LS-DYNA is a trademark of Livermore Software Technology Corporation.
- ACIS is a registered trademark of Spatial Technology, Inc.
- Parasolid and Unigraphics are registered trademarks of Electronic Data Systems Corporation.
- Windows, Windows NT, Windows 95, and Windows 98 are registered trademarks of Microsoft Corporation. Portions of the software contained on your FEMAP CD are copyrighted by Microsoft Corporation.

Other brand or product names are trademarks or registered trademarks of their respective holders.

- Portions of this software are copyrighted by Spatial Technology, Inc., Electronic Data Systems, Inc., INRIA, Cypress Software Inc., and Microsoft.



Manual Conventions

This manual uses different fonts to highlight specific features, to identify data that Windows will display, or input that you must type.

<code>Enter, Alt</code>	Shows one or more keys that you should press. In some cases, you will see combinations like Alt+Shift+Back-space. The plus signs show that you should press all keys simultaneously.
<code>a:setup</code>	Shows text that you should type. This is very similar to the keystrokes described above, but is typically used for strings of letters and/or numbers. The keystrokes typically refer to the more specialized, non-alphanumeric keys.
<code>OK, Cancel</code>	Shows text that you will see displayed by FEMAP in a dialog box control, or in the menu.
<i>heading</i>	Used for headings or titles of sections of the manual. Larger characters of the same style (or italics) are also used depending upon the nature of the section being introduced.
<code>text</code>	Used for all other normal manual text.

Throughout this manual, you will see references to Windows. Windows refers to Microsoft® Windows NT, Windows 95 or Windows 98. You will need one of these operating environment to run FEMAP for the PC. This manual assumes that you are familiar with the general use of the operating environment. If you are not, you can refer to the Windows User's Guide for additional assistance.

Similarly, throughout the manual all references to FEMAP, refer to the latest version of our software.

1 New Features

1.1 FEMAP Enterprise7-1
1.2 Updated Interfaces7-2
1.2.1 ABAQUS Interfaces7-2
1.2.2 ANSYS Interfaces7-2
1.2.3 I-DEAS Interfaces7-2
1.2.4 LS/DYNA Interfaces7-2
1.2.5 NASTRAN Interfaces7-2
1.2.6 UAI/NASTRAN Interface7-2
1.3 Picking and Common Dialogs7-2
1.3.1 Standard Coordinate Dialogs7-2
1.3.2 Entity Selection7-3
1.4 File Menu Commands7-3
1.4.1 Printing7-3
1.4.2 File Preferences7-3
1.4.3 Sketching7-3
1.5 Geometry7-3
1.5.1 Surface Creation7-3
1.6 Model Menu Commands7-3
1.6.1 Model Material7-3
1.6.2 Model Function7-4
1.7 Meshing7-4
1.7.1 Mesh Extrude Element7-4
1.7.2 Meshing Surfaces7-4
1.7.3 Mesh Points on Surface7-4
1.7.4 Multi-Surface Meshing7-4
1.7.5 Meshing Solids7-5
1.8 Modify Menu Commands7-6
1.8.1 Modify Project Mesh Onto Solid7-6
1.8.2 Modify Move By Offset Element7-6
1.8.3 Modify Renumber Materials7-6
1.8.4 Modify Color7-6
1.8.5 Modify Update Elements Reverse7-6
1.8.6 Modify Update Elements Split Quads7-6
1.9 List Menu Commands7-6
1.9.1 List Loads7-6
1.10 Views and Graphics7-7
1.10.1 View Rotate and Align7-7
1.10.2 Free Face and Hidden Line7-7
1.10.3 Graphics.7-7
1.10.4 Render Mode7-7
1.10.5 Dynamic Rotation7-7
1.11 PostProcessing7-8
1.11.1 Contour Postprocessing.7-8
1.11.2 Beam Diagram Postprocessing7-8
1.11.3 Contour Vector Postprocessing7-8
1.11.4 Section Cut Postprocessing7-8
1.11.5 Freebody Postprocessing7-8



New Features



This document includes updated information for the documentation provided with FEMAP Version 7.1. The information in this document and in the Help system is more up-to-date than that in the manuals. Also check our web site <http://www.femap.com> for additional information.

1.1 FEMAP Enterprise

Version 7.1 represents the first release of FEMAP Enterprise. This new product contains a broad range of new geometry interfaces, including:

CATIA Import - Provides the ability to read CATIA model files, and Express files from CATEXP. Catia model files must be from CATIA v4.1.x or v4.2

IGES Import - in addition to the basic IGES interface that is included in FEMAP and FEMAP Professional, an advanced IGES interface that supports many additional entity types is included in FEMAP Enterprise. This supports IGES Standards 4.0 to 5.3.

VDA Import - this interface provides direct access to VDA files up to v2.0

IDEAS Import - Provides access to IDI files generated by I-DEAS MS8. Currently only part (not assembly) geometry is supported, but this provides a very fast, very robust link between FEMAP and I-DEAS

PRO/E Import - Provides direct access to model files from PRO/ENGINEER v16 to v20. From other versions of PRO/E, the IGES interfaces will continue to provide very robust geometry transfer.

Solid Edge Import - Direct access to Parasolid geometry in Solid and Sheet Metal part files.

Unigraphics Import - Direct access to Parasolid geometry from Unigraphics v11 thru v15

ACIS and Parasolid Import - Both of these interfaces provide the ability to convert the imported geometry to the other solid modeling engine. If you normally, work with Parasolid, but you have some ACIS geometry that you need to use, this lets you convert it to Parasolid so that it can fully interact with your other geometry.

IGES Export - in addition to advanced IGES import, it is now possible to export Parasolid geometry to IGES format.

1.2 Updated Interfaces

1.2.1 ABAQUS Interfaces

An option has been added to use the entity titles as the titles for SET commands. Added support for the *INCLUDE command and INPUT= options on various commands for the model read interface.

1.2.2 ANSYS Interfaces

Corrected a problem with the TSHAP command when writing triangular contact elements. Corrected a problem with the TBPT command for stress-strain curves on non-linear plastic materials.

1.2.3 I-DEAS Interfaces

Added an interface to read I-DEAS part geometry from the IDI (Interoperability Data Interface). Significantly improved and updated the I-DEAS Universal File interfaces to include more model data, including some basic capabilities to read nodal results and solid element stresses.

1.2.4 LS/DYNA Interfaces

Corrected a problem that caused the Springback and Implicit solver options to be reversed. Also improved support for postprocessing of some older versions of DYNA by providing an option that allows you to force each new step to start in a new file.

1.2.5 NASTRAN Interfaces

Significantly improved the performance of writing NASTRAN model files that have both a large number of enforced displacements and a large number of constraints (25000 or more each). Added warnings if you attempt to write laminates without a material orientation, or if you specify plate offsets but then translate to a plate type that does not support them.

1.2.6 UAI/NASTRAN Interface

Stopped writing MAT4 and MAT5 cards which were written incorrectly for UAI/NASTRAN and were not required for the structural analysis types that the UAI interface supports.

1.3 Picking and Common Dialogs

1.3.1 Standard Coordinate Dialogs

The "In Workplane" coordinate definition dialog now properly switches labels as you toggle between polar and rectangular coordinates.

1.3.2 Entity Selection

If you press the Alt-key while picking, and you are in either normal selection mode, or "Front" selection mode, you will switch to Query Selection for that pick. This eliminates the need to manually change your selection when you have a few picks where query is needed. Added a Preview command to the Pick menu that shows the entities that have been selected.

1.4 File Menu Commands

1.4.1 Printing

Corrected a problem that occurred when you tried to print headers or footers, or a multiple window layout on certain DeskJet printers. These printers do not properly reset the clipping region during printing resulting in certain portions of the print being lost. An option has been added to File->Page Setup to manually reset the Clipping Region that will overcome this problem. Added an option to print directly to a file instead of your printer.

1.4.2 File Preferences

Added a View Preference to use (or not use) Vertex Arrays in Render Mode. Vertex arrays are a feature of OpenGL that, if properly supported by your graphics card and drivers, allows improved graphics performance. Unfortunately, we have found cards that claim to support this feature that do not work properly when it is enabled.

1.4.3 Sketching

Corrected a problem that caused FEMAP to crash if you attempted to do a File Exit while you were still in Sketch Mode.

1.5 Geometry

1.5.1 Surface Creation

Changed the way that surface are created by the Surface->Corners and Surface->Ruled commands when using the ACIS geometry engine. These surfaces now have their parametric directions created consistently with the other geometry engines. Also updated the height and width specifications for the Geometry->Surface->Planar command for the ACIS engine, so that it was consistent with the other engines.

1.6 Model Menu Commands

1.6.1 Model Material

User defined materials can not have text fields with no input field. Simply define the field as Axxx (xxx is any number) in the material definition file to create the text without an input field. Added a Fluid Material Type, additional phase change properties and support for ThermoOptical properties on some other standard material types.

1.6.2 Model Function

Added the capability to automatically create periodic functions by duplicating a range of values.

1.7 Meshing

1.7.1 Mesh Extrude Element

Improved the "Normals with Thickness Correction" method to better account for situations with multiple elements coming together at a node - for example at the corner of a box.

1.7.2 Meshing Surfaces

Corrected a problem that caused a mesh on a Boundary that was mapped to a surface to come out at the origin, rather than on the surface, if you smoothed the mesh.

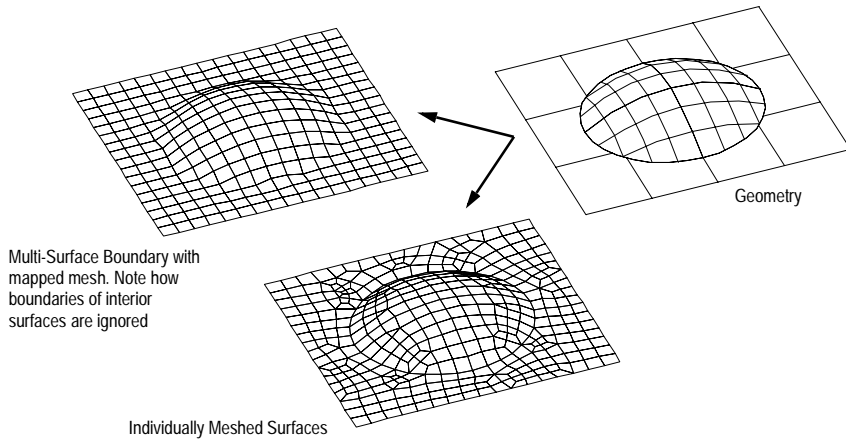
1.7.3 Mesh Points on Surface

The Mesh Points on Surface command has been updated so that you can easily create new mesh points, use existing points, connect to existing nodes in another mesh, or delete the points that you have already assigned.

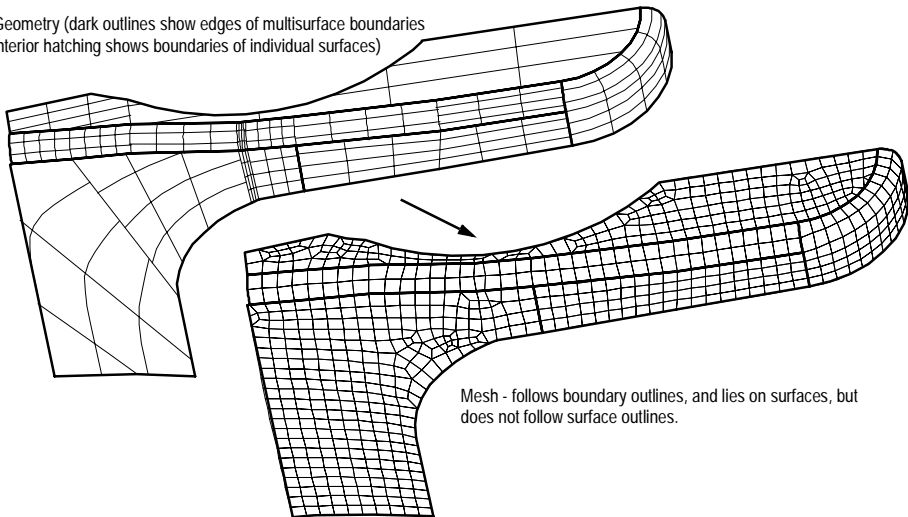
1.7.4 Multi-Surface Meshing

Whether it is called Multi-Surface Meshing, Compound Surface Meshing, Section Meshing, or some other name, this tool provides the ability to consider multiple surfaces as one and mesh across the combined group of surfaces as if they were a single surface. Interior boundaries are simply ignored. Several new commands have been added to facilitate this operation. Geometry->Boundary Surface->From Surfaces on Solid lets you quickly create a boundary surface by picking multiple adjacent surfaces from a stitched solid. Geometry->Boundary Surface->Update Surfaces provides automatic updating of boundary surfaces that were created with the "From Surfaces on Solid" command, if the underlying solid geometry has changed. Geome-

try->Boundary Surface->Edit Surfaces allows you to edit the definition of one of these Boundary Surfaces if you want to add or remove surfaces.



Geometry (dark outlines show edges of multisurface boundaries
interior hatching shows boundaries of individual surfaces)



1.7.5 Meshing Solids

The solid meshing commands will search for a solid property and select it as the default if the active property is not a solid property.

1.8 Modify Menu Commands

1.8.1 Modify Project Mesh Onto Solid

Allows you to take a mesh and project it onto an entire solid or onto a group of surfaces from a solid. This can be used to mesh a planar outline of a region and then move the mesh onto the solid.

1.8.2 Modify Move By Offset Element

This command provides the capability to move elements along a vector, or along their element normals. Moving planar elements along their normals can be a simple way to create a midplane mesh of constant thickness parts - simply mesh either the inner or outer surfaces, and move the elements along their normals by one half of the thickness.

1.8.3 Modify Renumber Materials

Corrected a problem that caused function references on materials to be lost if you renumbered the material.

1.8.4 Modify Color

Added the ability to randomize surface colors. Changed the default color of a boundary surface to be different than the normal surface color - this is intended to make the boundaries created by the new multi-surface meshing commands visible when displayed on top of the underlying surfaces.

1.8.5 Modify Update Elements Reverse

Previously when you reversed element directions, loads applied to the element faces would flip direction. This could also happen for axisymmetric elements that were built in an incorrect modeling plane and flipped automatically as you translated your model. Now the loads are properly adjusted to maintain their direction when the elements flip.

1.8.6 Modify Update Elements Split Quads

Corrected a problem that left the midside nodes of parabolic quads as nondeletable after you split them into triangles. Previously you had to run File Rebuild before you could delete these nodes.

1.9 List Menu Commands

1.9.1 List Loads

Significantly improved the speed of listing loads from sets that have a very large number of nodal or elemental loads. This is most evident in load sets with 20,000 loads or more.

1.10 Views and Graphics

1.10.1 View Rotate and Align

The View Rotate command has been changed to a submenu. The old View Rotate command can now be accessed via the View->Rotate->Model command. The same shortcut keys Ctrl+R or F8 still work as before. View->Align By->Dynamic has been moved to the new View Rotate menu, and new commands have been added to allow you to pick the dynamic rotation center and axis.

1.10.2 Free Face and Hidden Line

Significantly improved the performance of the computation of free faces in large solid models. This greatly improves the display speed on View Regeneration for Free Face and all Hidden Line displays outside of Render mode. It also improves performance of other commands that require free face information - like picking free faces. Also corrected a problem in the usage of the "Free Edge and Face" View Option. Previously the Use Midnodes and Skip Midnodes options worked backwards for Free Face and Hidden displays.

1.10.3 Graphics

Corrected a problem that caused all graphics to disappear except the axes, origin and workplane. The problem was created if you closed a view within a couple of seconds from the time when you resized it. Previously the only ways out of this problem were to exit and restart FEMAP, or create a new view and resize it.

1.10.4 Render Mode

Corrected problems that occurred on several graphics boards/drivers. This includes problems with the GeForce cards (requires disabling Vertex Arrays in File Preferences).

1.10.5 Dynamic Rotation

The dynamic rotation command has been improved to allow dynamic rotation about a location other than the center of the screen, and about any vector/axis that you choose. New commands have been added to the View Rotate submenu to allow you to define these options, or you can define them from a new Options menu in the dynamic rotation dialog box.

Dynamic rotation can now be done even if you have both curves and elements turned off for normal displays, as long as you have one or both of them in your model.

1.11 PostProcessing

1.11.1 Contour Postprocessing

Corrected a problem that gave invalid contours if you used elemental contouring on a plate model, and you chose either Max or Min data conversion (not averaging), and you turned on the display of plate thicknesses.

1.11.2 Beam Diagram Postprocessing

Corrected a problem that occurred if you set the beam diagram direction to Elemental Y or Z, and you displayed the diagram on a deformed model, with a deformation scale that was large enough to cause the deformed beam X axis to pass through the element orientation vector. In this case the direction of the diagram was previously reversed - it is now drawn properly. Added support for beam diagrams of Beam Equivalent Stress, Beam Plastic Strain, Beam Total Strain and Beam Creep Strain when these values are recovered from your analysis.

1.11.3 Contour Vector Postprocessing

Corrected a problem that occurred on uniaxial solid models with solid elements that had 2 zero shear stresses. If you plotted a standard Max Principal Stress contour vector, the vectors on those elements was zero. It is now computed properly.

1.11.4 Section Cut Postprocessing

Added support for elemental contouring in a section cut display. Previously if you selected elemental contours with section cuts, all contour values were displayed as zero.

1.11.5 Freebody Postprocessing

If you have multiple output sets, some with Freebody output, and others without, when you went into the Freebody dialog box, the graying was not handled properly unless you had selected the output set in a previous command. This problem has been corrected.