Marc® and Mentat® 2008 r1

Release Guide

MSC Software

Corporate

MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707
Telephone: (800) 345-2078
FAX: (714) 784-4056

Europe

MSC.Software GmbH
Am Moosfeld 13
81829 Munich
GERMANY
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Asia Pacific

MSC.Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Fax: (81) (3)-6911-1201

Worldwide Web

www.mscsoftware.com

MA*V2008R1*Z*Z*Z*DC-REL

Contents

Marc and Mentat 2008 r1 Release Guide

Overview

The release of Marc 2008 r1 family of products broadly encompasses the following objectives:

- Major new enhancements in several areas in both solver and User Interface capabilities
- Substantial increase in robustness of analysis
- Improvements in quality – several defects in the previous versions have been fixed
- Computational improvements

List of Latest Functionalities

This Marc 2008 r1 release includes several enhancements as well as defects corrected since the previous MSC.Marc 2007 r1 release. A summary of the latest features for both the solver and graphical user interface is given below and the complete details of these features are contained in the Marc 2008 r1 Marc Volumes A through D.

Marc Functionalities

Element Technology

- Element Type 202 3-D 15-node Pentahedral
- Element Type 203 3-D 15-node Pentahedral (Heat Transfer and Electrostatic)
- Element Type 204 3-D 6-node Magnetostatic Pentahedral
- Element Type 205 3-D 15-node Magnetostatic Pentahedral
- Element Type 206 20-node 3-D Magnetostatic Element

Material Behavior

- TSHEAR now includes a parabolic distribution of transverse shear strain for Solid Shell and 3-D Composite Brick Elements (Types 149 and 150).
- Mixture models now allow the user to define a composite material that consists of multiple components for membrane, shell, and continuum element types. Effective material properties are formed based upon the volume fraction of each material component. There are three variations of the mixture model which have consequences on the type of material that may be used in each component. The component material is defined using the conventional ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, MOONEY, etc. options. The mixture model acts as a continuum in the sense that debonding between the components is not considered in any of the models.
- Shape memory has been enhanced to allow different elastic properties for the Austenite and Martensite phases when using the mechanical (Auricchio) model.
- The UVSCPL user subroutine now automatically calculates a tangent stiffness matrix for the implicit formulation of viscoplasticity.

- Combined isotropic and kinematic hardening can now vary the amount of kinematic hardening allowed.
- Material orientation has two new options: 3D LOCAL and CURVE. The 3D LOCAL option allows the orientation to be defined using a local element system for hexahedral elements together with three rotations. The CURVE option uses a set of NURBS curves for defining the orientation. The first material direction is defined using the tangent of the closest point of the closest curve. It is supported for 2-D elements, shells, solid shells and 3-D composite solids.
- The HYPOELASTIC option now allows you to select the configuration used for the strain evaluation.
- The simplified damage model has been improved. Note that the call list for the UDAMAG user subroutine has changed and is not backward compatible.

Assembly Modeling

- Pin connections – (See PIN CODES or GEOMETRY option)

Loads and Boundary Conditions

- If multiple parts are rotating, then different rotation axes may be defined for centrifugal and Coriolis loads.
- Cyclic loading can be done either with a table that repeats the load or a single load case that is repeated by using the BEGIN SEQUENCE and END SEQUENCE history definition options.

Procedures

Mesh Splitting

- Marc now has the capability to automatically split up the mesh during the analysis. The mesh split can be done at nodes and along element edges (2-D and shells) or element faces (3-D). A new node is created where the split is done. This node inherits the current properties of the original node. The elements at one side of the split are modified to use the new node to create the opening in the mesh.
- An option is available in the DELAMIN option for inserting interface elements using a cohesive zone material model where the mesh is split. This is automatically done whenever a split takes place. This supports all available interface elements.
- A new user subroutine, USPLIT_MESH, can be used for splitting up the mesh along edges/faces.

Fracture Mechanics

- Virtual Crack Closure Technique (VCCT) for evaluating energy release rate and stress intensity factors has been extended to include four crack growth criteria.
- An option has been introduced so that a crack can grow along element edges for 2-D and shell elements. The element edge closest to the crack growth direction is used. Using the procedure implemented for mesh splitting, new nodes are automatically inserted and the element connectivity is changed for elements around the crack tip in order to grow the crack. The new nodes inherit the properties of the original parent nodes. The crack growth direction can now also be specified to be along a pure crack mode or along a user defined vector.

Adaptive Meshing

- The local adaptive meshing criterion “node within a box” has been extended to a cylinder and sphere.
- Local adaptive meshing has been extended to electrostatic and magnetostatic analyses.

Contact

- It is now possible to break up the glued connection between contact bodies using a stress criterion. When a node is released due to breaking glued contact, it changes status from being glued to standard contact permitting separation and friction. The contact stresses are calculated using extrapolated stresses for solid elements and as contact force divided by equivalent area for shell elements. Nodal post codes are available for postprocessing the separate terms in $\left(\frac{\sigma_n}{S_n}\right)^m + \left(\frac{\sigma_t}{S_t}\right)^n > 1$ as well as the sum. A nodal post code for the deact glue status is also available.
- Shell edge-to-edge contact is now possible and requires switching on the beam-to-beam contact flag on the CONTACT option and, for a shell contact body combination, the edge-to-edge contact flag on the CONTACT TABLE option. For edge-to edge contact, half of the thickness of the shell is used to set the contact radius.
- Bilinear friction model is available for beam-beam contact.
- Variable beam radius is taken into account for beam-beam contact.
- One can evaluate the distance to contact and display it.

Deactivation

- An option is available for deactivating islands of disconnected elements and deactivating elements that go inside out. It is typically used together with crack growth, mesh splitting, or progressive failure analysis with element deactivation to avoid problems with rigid body motion of the islands that no longer are needed in the analysis.

Dynamics

- A generalized alpha operator has been added for dynamics. It allows convenient setting of the spectral radius to be used for a given problem, with low spectral radii closer to 0 recommended for contact problems and high spectral radii closer to 1 recommended for non-contact problems.

Mechanical Wear

- Mechanical wear is an important physical phenomena in any structure subjected to repeated loadings. Several wear models along with a user subroutine, UWEAR, have been added to simulate mechanical erosion on a surface that is being contacted. This can be applied to manufacturing, disk brakes, bearings, gears, tires, and seals.

Heat Transfer

- Boundary conditions (INITIAL TEMP, FIXED TEMP, and POINT FLUX) on the degrees of freedom of heat transfer shells is easier to apply. These boundary conditions may also be entered as a function of the thickness.
- The CONM1 option has been enahnced to support thermal capacitance.
- The viewfactors associated with radiation can be symmetrized.

Capacitance Calculation

- A conductor in an electrostatic analysis can be defined as an electromagnetic body using the THERMAL CONTACT model definition option. No two conductors can touch each other. A subset of the conductor bodies can be considered for capacitance computation using the EMCAPAC history definition in the THERMAL CONTACT model definition option.

Resistance Calculation

- A conductor in a Joule heating analysis can be defined as an electromagnetic body using the THERMAL CONTACT model definition option. Two conductors can touch each other but cannot overlap. If two conductors touch, it implies flow of current between the two. A subset of the conductor bodies can be considered for a resistance computation using the EMRESIS history definition option.

Electromagnetics

- A Backward-Euler time stepping scheme has been added for transient electromagnetic analysis. The new formulation works better than the existing Newmark-Beta algorithm for low values of permittivity and low frequency loading. The Newmark-Beta algorithm has been improved and works well when permittivity-related terms are significant in the transient response.

Superelement/DMIG Support

- The older method of susbtructure/superelement using the NEWDB, SUBSTRUC, and SUPERIN options is no longer supported.
- The procedure using the SUPERELEM, DMIG, and K2GG has been substantially improved. This includes:
  - Faster creation of superelements.
  - Alternative DMIGB file format which is substantially smaller.
  - The use of DMIG in large deformation problems where the DMIG stiffness undergoes large rotation.

Computational Improvements

- Assemble improvements - 20-40% reduction is assembly time.
- Improvements to superelement creation.
- Contact speed improvements with complex NURBS.
- Reduced back-up memory requirements.
- Improved Marc iterative solver (2) when Lagrange multipliers are used.
- Fluid-solid added mass approach.

Output

- New Post Codes.
- SORT NODE has been expanded to sort by magnitude of vector quantity.
- Control of output of SPRING results using PRINT SPRING..

Mentat Functionalities

Support of new Marc functions:

- DELAMIN support
- CONM1 and CONM2 for generalized lumped mass
- The option to use a control node for VOLUME FLUX.
- Pin connections – (PIN CODES)
- Heat transfer boundary conditions for shell element.
- Mixture material model.
- Extensions to Auricchio shape memory.
- Extensions to HYPELA.
- EMPRESIS.
- EMCAPAC.
- Extersions to CONTACT TABLE for shell edge contact.
- Wear analysis via RECEDING SURFACE.
- ACTUATOR with tables.
- ORIENTATION (CURVE and 3D LOCAL).
- Radiation view factor enhancement.

General

- On Windows, data that is copied from the clipboard can be pasted into Mentat using Ctrl-V. If the cursor is in a text field (such as the geometric property name, or the area of a truss), the data is pasted into this field and can then be applied using the Enter key. If the cursor is not in a text field, the data is pasted to the "terminal" dialog area and processed through the command processor.
- On Windows, data can be copied from Mentat to the clipboard. If the cursor is in a text field (such as the geometric property name, or the area of a truss), Ctrl-C will copy the current text of this field to the clipboard. If the cursor is not in a text field, Ctrl-C will copy the current command line buffer to the clipboard, and Shift-Ctrl-C will copy the entire contents of the command buffer to the clipboard.
- The Customer Entitlement ID is now shown in the HELP>THIS VERSION popup menu.
- When defining a node path for a PATH PLOT, the nodes of the resulting path are now automatically flagged as SELECTED. This will facilitate checking the defined path.

- Individual free nodes not part of mesh or links can be made invisible now. Individual free points not part of a curve or surface can also be made invisible. In previous Mentat versions, the visibility of nodes and points was fully controlled by the visibility of the mesh and geometry parts and free nodes and points where always visible.
- New commands *`visible_nodes`, *`invisible_nodes`, *`visible_points`, and *`invisible_points` are available (in PLOT>MORE) to change the visibility of free nodes and free points. In addition, the *`visible_selected` and *`invisible_selected` commands (MAKE VISIBLE/INVISIBLE in SELECT menu) now also change the visibility of the selected free nodes and points and *`visible_all` and *`invisible_all` also change the visibility of all the free nodes and points.
- The visibility of nodes which are part of an element, link or RBE and the visibility of points which are part of a curve or surface is still controlled by the visibility of the element, link, curve or surfaces as before. Such nodes and points cannot be made invisible by the new commands.
- This may cause existing procedure files to run differently. The old visible behavior can restored by executing the command

  `*prog_option compatibility:prog_version:ment2007` at the start of such a procedure file.
- New commands *`visible_free_nodes`, *`invisible_free_nodes`, *`visible_free_points` and *`invisible_free_points` are available to control the visibility of all the free nodes or free points of the model. This is particularly useful for the SWEEP commands, if only part of the mesh is visible.
- The default method for generation of PostScript files has been changed from RASTER to VECTOR. Older procedure files may have to be adapted for this.
- A script `mdl2mdl` has been added in the bin directory to run the `mdl2mdl.exe` utility on platforms that use compiler related shared libraries.

Preprocessing

- Sweep option has been enhanced.
- Solid beam sections can now be displayed on the model. So far, this was only available for the thin-walled section beams (circular and general). Now, for solid section beams, all possible sections can be drawn, except for elliptical, which will be covered in a subsequent release. Note that for a solid section beam with "entered" properties, currently a solid circular section is drawn with a diameter derived from the area.
- The option to use a control node for a VOLUME FLUX is now supported. For EDGE FLUX and FACE FLUX, this option was already supported.
- New commands have been added to clear the assigned geometric property for a list of elements, curves or surfaces. The buttons are located in GEOMETRIC PROPERTIES>TOOLS. Note that the REMOVE UNUSED GEOMETRIES button has been moved to the TOOLS submenu.

- The button DRAW X-Y AXES in the BEAM PLOT SETTINGS menu and the BEAM DIAGRAM SETTINGS menu has been renamed to DRAW LOCAL AXES. A button has been added to access the BEAM PLOT SETTINGS menu from the GEOMETRIC PROPERTIES>MECHANICAL PLANAR menu.
- For NODAL TIES, SERVO LINKS, SPRINGS/DASHPOTS, RBE2s, RBE3s, and RRODs, the current entity can now be selected graphically using the PICK button.
- A new command FROM CVS (`*from_curve_to_table`) has been added to generate tables from a list of curves in the model. The tables represent X(t), Y(t) and Z(t). The number of CURVE DIVISIONS of the curves define the number of table points. This option can be used to prescribe a position as function of time, when the motion is known in the form of a curve.
- The menus for post file revision 13 in JOBS and LOADCASE have been redesigned.
- A command has been added in the CHANGE CLASS menu to change collapsed HEXA elements into PENTA elements.

Marc Input File Writer

- Either POST or POST INCREMENT is now written for a loadcase and no longer both. The POST option is written if the post file revision is 13 and the post data (variables or elements/nodes/bodies) have changed. The POST INCREMENT option is written if only the frequency has changed.
- The 12th field of the POST model definition option (trial solutions on post file) is now fully supported. This flag can be set via

  ```
  *job_option post_trial:off -> 0
  *job_option post_trial:on -> 1
  *job_option post_trial:loads -> 2
  *job_option post_trial:contact -> 3
  ```

  Until now, only on and off were supported. There are no menus for this flag.

Postprocessing

- In addition to MPEG and AVI movies, animated GIF files can now be selected for generation with Windows. Animated GIFs can be imbeded into Microsoft power point presentations, whereas MPEG and AVI cannot.
- A toggle button, LEGEND BACKGROUND, has been added to allow putting a background behind the text of the legend. This may be helpful to read the legend when it is in front of the model. The color index used for the background is 4, the same as used for GRAPH FILL.
- The colors of the PATRAN CONTOURMAP have been updated to match those of the current Patran version.

- The PATH-PLOT menu has been enhanced to allow path plots along a list of sample points. The sample points are generated by clicking a curve and entering the number of divisions for the curve. The points are mapped onto the mesh and the results are interpolated to the sample points. An option has been added to automatically clean AVI and MPEG movie animation files.

Graphics

- New visibility scheme for free nodes and free points.

Marc Reader

- Support has been added for the thermal boundary condition options that have been introduced in 2007 R1, such as the CONTROL NODE data, LOAD TYPE: DIRECTED (Marc input option QVECT) and the EVALUATION TEMPERATURE method for films.
- Reading of the history definition (loadcase information) of a Marc input file from Mentat is now possible for new style table input files. The most important loadcase data are read, including:

Marc Input File History Definition Options Read with Mentat Reader	
Supported	**Not Yet Supported**
ACTIVATE	ACTUATOR
ADAPT GLOBAL	BEGIN SEQUENCE
ADD RIGID with TABLES	DAMPING COMPONENTS
ANNEAL	DMIG-OUT
APPROACH	END SEQUENCE
ASSEM LOAD	GAP CHANGE
AUTO CREEP	MOVE
AUTO INCREMENT	
AUTO LOAD	
AUTO STEP	
AUTO THERM	
AUTO THERM CREEP	
BUCKLE	
CHANGE RIGID	
CONTACT NODE	
CONTACT TABLE	
CONTINUE	
CONTROL	

Marc Input File History Definition Options Read with Mentat Reader	
Supported	**Not Yet Supported**
CREEP INCREMENT	
DEACT GLUE	
DEACTIVATE	
DYNAMIC CHANGE	
EXCLUDE	
GRID FORCE	
HARMONIC	
INCLUDE (all data lines imported)	
INERTIA RELIEF	
LOADCASE	
MODAL SHAPE	
PARAMETERS	
POST	
POST INCREMENT	
RECOVER	
RELEASE	
SPECTRUM	
SS-ROLLING	
STEADY STATE	
SUPERELEM	
TIME STEP	
TITLE	
TRANSIENT	
VCCT	

DXF Reader

- An option has been added to select if entities are to be imported from MODEL SPACE only, PAPER SPACE only, or BOTH. This option is also available as a command line option (-s 0/1/2) when the DXF reader is run outside of Mentat. Mentat 2007 R1 imported from BOTH, previous versions imported from MODEL SPACE only.

Nastran BDF Reader

- The parameters and options are now translated from the BCBODY option. The name of the resulting rigid contact body may be derived from the comments in the BDF-file.

STL Reader

- Multiple STL files can now be translated when the reader is run outside of Mentat.
- An option has been added to select FEM or GEOMETRIC entities (elements or surfaces) as TARGET ENTITIES of the import operation.

STL Writer

- It is now possible to export TRIA6, QUAD6, QUAD8 and QUAD9 element faces. The midside nodes rae ignored, and quadrilateral faces are exported as pairs of triangular faces.

Demonstration Problems

In addition to these latest functionalities the *Marc User's Guide*, Marc's *Volume E: Demonstration Problems* demonstrates a wider set listed below. Cross-reference Tables in Chapter 1 of Volume E list the options used in these new demonstration problems.

Chapter. Problem	Title
2.86	Demonstration of Multiple Rotation Axis for Spinning Cylinders
2.87	Example of Elastic Mixture Model
2.88	Using a Curve to Define Material Orientation
3.47	Cyclic Plasticity
4.25	Modeling Revolute-Translational Joint with PIN CODE
8.20	Eigenvalue Analysis of a Ribbed Plastic Cover
8.21	Composite Material Orientation Defined by Curve
8.22	Nonlinear Simulation of a Mixture Material
8.23	Gear Analysis Using Substructures
8.24	Composite Delamination Analysis of a 3D Block
8.28	Prediction of Tool Wear in a Metal Cutting
8.29	Coupled Simulation of Mechanical Wear in 3-D
8.30	Fiber Pullout Using the Breaking Glue Option
12.7	2-D Electrostatic Analysis: Concentric Cylindrical Capacitor
12.8	2-D Electrostatic Analysis: Two Parallel Cylinders Capacitor
12.9	2-D Electrostatic Analysis: Tapered Capacitor
12.10	2-D Electrostatic Analysis: Parallel Plate Capacitor Bi-layered Dielectrics
12.11	2-D Electrostatic Analysis: Charged Conducting Sphere

Chapter. Problem	Title
12.12	3-D Electrostatic Analysis: Charged Conducting Sphere
12.13	3-D Electrostatic Analysis: Two Charged Conducting Spheres
12.14	3-D Electrostatic Analysis: Two Concentric Charged Conducting Spheres
12.15	2-D Electrostatic Analysis: Parallel Plate Capacitor with One Plate Grounded
12.16	2-D Electrostatic Analysis: Parallel plate Capacitor with Both Plates Charged
12.17	3-D Electrostatic Analysis: Parallel Plate Capacitor with Volume Charge
12.18	3-D Electrostatic Analysis: Parallel Plate Capacitor with Surface Charge
12.19	3-D Electrostatic Analysis: Point Charge in Free Space
12.28	2-D Magnetostatic Analysis: Axisymmetric Solenoid
12.29	2-D Magnetostatic Analysis: Planar Coaxial Cable
12.30	2-D Magnetostatic Analysis: Straight Current Sheets
12.31	3-D Magnetostatic Analysis: Straight Current Sheets
12.32	3-D Magnetostatic Analysis of Straight Infinite Line and Sheet Currents
12.35	2-D Axisymmetric Harmonic Electromagnetic Analysis of a Long Wound Solenoid in Free Space with Specified Currents
12.36	2-D Axisymmetric Harmonic Electromagnetic Analysis of a Long Wound Solenoid in Free Space with Specified Point Currents and Varying Frequency
12.37	2-D Planar Electromagnetic Harmonic analysis of a Coaxial Cable with air inside
12.38	Harmonic Electromagnetic Analysis of a Wave Guide
12.39	Transient Electromagnetic Analysis Around a Conducting Sphere
12.41	Electromagnetic Analysis of an Infinite Wire
12.42	Capacitance matrix computation of two conductors placed between two grounded parallel long plates using 2-D Electrostatics
12.43	Resistance computation of five conductors of equal size placed in tandem using 2-D Joule heating
12.44	Contact in Magnetostatics

List of Corrected Defects in this Release

Marc

Adaptive Meshing and Rezoning

1 When using the penetration check for self-contact in a 2-D analysis, the outline of a newly generated mesh could be incorrect.

2 When manual remeshing was used for elements 155 (3+1-node plane strain element), 156 (3+1-node axisymmetric element) and 157 (4+1-node tetrahedral element), the results were wrong if not all the coordinates are given in the COORDINATE CHANGE option.

3 If the CYCLIC SYMMETRY option was used together with automatic global remeshing, the analysis would prematurely stop if the remeshed body did not have multipoint constraint equations following from cyclic symmetry conditions.

4 If remeshing is combined with element deactivation, the new mesh would be incorrect if remeshing occurs immediately after the element deactivation.

5 If the QVECT load option is combined with global remeshing, the program could crash or abort with exit,1055.

6 If global remeshing is performed on a contact body containing beam elements, the program could crash instead if giving an error message.

7 In rare cases, the program could crash in the shaver mesher.

8 The INSERT option did not function properly in combination with local adaptive remeshing of the host elements.

9 The program could crash if the POINT FLUX option is used in a heat transfer analysis using local adaptive remeshing.

Constraints, Boundary Conditions and Load Stepping

1 A global-local analysis could stop with exit,13 if the global analysis used shell elements and the local analysis used brick elements.

2 The minimum time step in an analysis using quasi-static inertial damping (flagged by a 1 on the 10th field of the 2nd data block of the AUTO STEP option) was not working properly in conjunction with Marc post file version 2000 and earlier versions.

3 The constraints following from the CYCLIC SYMMETRY option could be incorrect for a modeled part spanning 180 degrees.

4 The INITIAL TEMP option did not correctly support cases where nodes have more than eight degrees of freedom per node.

5 A model using TRANSFORMATION and PRE STATE could get incorrect initial temperatures.

6 When using the FOUNDATION or FILMS option with old style input, the reassembly of the operator matrix (1) did not work for a linear elastic analysis flagged by the ELASTIC parameter option and (2) took always place for a regular analysis, even if foundation or film information did not change.

7 When the AUTO STEP option is used with damping energy based time step selection, then the solution could be wrong if the initial damping energy cannot be properly defined. This might e.g. happen in cases where the solution of the first increment of the AUTO STEP load case only shows nearly zero displacements following from projecting nodes on a contacted body.

8 Point loads using the automated follower force setting (FOLLOW FOR option) did not work properly in the following cases: (1) with old style input, when the load vector remained unchanged for a part of the analysis and then changed its direction; (2) with new style input, when the arc length method (AUTO INCREMENT option) was used.

9 In a global-local analysis, the local displacements could be wrong if the global analysis used contact with stress-free projection.

10 The internal program flag to reassembly the operator matrix was not set correctly for a transient dynamics or a transient heat transfer analysis in conjunction with the AUTO STEP option. This affected linear runs when the starting time step for a new loadcase is different from the previous time step.

11 The applied surface heat flux for element 157 (4+1-node tetrahedral element) was incorrect.

12 Tying types 32 and 34 were incorrect when used with higher order Herrmann elements.

13 The coordinates used with tying types larger than 1000 (to get the updated coordinates) could be wrong, especially in cases where multiple element types are present.

14 The internal nodal code of spring nodes was incorrect for springs not being attached to an element, which could deteriorate the convergence behavior.

15 The Coriolis and PLOAD4 forces were incorrect for element 136 (6-node pentahedral element).

16 With old style input, the RELEASE or RELEASE NODE option was not treated correctly if, in the same load case, the POINT LOAD option was placed after the RELEASE or RELEASE NODE option.

17 The SURFACE ENERGY option was not allowed to be used in a mesh containing shell elements. In version 2008 r1 this is allowed, as long as the surface energy is not defined on a shell element.

18 With old style input, DIST LOADS ibody types 106, 107, 108, 110, 111, 112 and 113 were wrong in a harmonic analysis.

19 When both the FILM and FOUNDATION options are used, the results would be incorrect.

20 PLOAD4 forces were not correctly adapted if the SCALE option is used.

Contact

1 When enforcing C^0-continuity via the SPLINE option in a 3-D contact analysis, the geometry of the internally generated Coons surface could lose some accuracy for segments containing an edge with a normal vector discontinuity.

2 If, in a 3-D contact analysis with local adaptive meshing. the SPLINE option was used and edges with a normal vector discontinuity were in the adapted mesh area, then the program could crash.

3 When optimized contact constraints were used in combination with initial stress free projection, the results could be inaccurate.

4 When optimized contact constraints were used in a self-contact problem with more than 100 isolated regions, the program could prematurely stop with exit,8888.

5 For nodes of shell elements, the contact normal vectors at the start of an increment were calculated based upon the shell mid-plane, but at the start of a cycle they were based upon the shell bottom-plane. This inconsistency could give problems in cases where the shell thickness varied significantly over adjacent elements.

6 The analytical representation of a deformable body in an mfd-file could be incorrect for 3-D quadratic elements.

7 When optimized contact constraints were used and a node was glued to multiple bodies, it could happen that in the final solution the glue conditions were not satisfied.

8 If a node is touching a deformable and a rigid body with glued conditions, the program could crash.

9 The procedure to update the outline of contact bodies would not be called if elements belonging to a contact body are activated and elements not belonging to a contact body are deactivated.

10 Elements 45 (3-node 2-D beam element) and 89 (3-node axisymmetric shell element) could give incorrect results in a contact analysis, since not all the relevant nodes and segments were included in the contact body description.

11 The approach velocity of a load-controlled rigid body was always in the global xyz-system instead of in the local system of the load-controlled body node.

12 If separation was based upon extrapolated integration point stresses, then for linear continuum elements the contact stresses on the post file were still derived from the force divided by the equivalent area.

13 A 3-D contact analysis with NURBS surfaces having zero area segments could stop with exit,9992 (message: `length=0.d0 in fol3d_proj`).

14 A contact analysis with quadratic elements and linearized contact in combination with local coordinate systems for nodes in the contact area gave incorrect results.

15 For dynamic contact problems, the dynamic penetration algorithm could sometimes predict an erroneous penetration factor of 1.0×10^{-6}.

16 In rare cases, a 2-D contact analysis with quadratic elements could stop with exit,2.

17 When the CONTACT option is used in a spectrum response analysis, the analysis would stop with exit,13.

18 If THERMAL CONTACT is used in a model having contact bodies consisting of linear continuum elements and bodies consisting of quadratic continuum elements, the program could crash.

19 In a coupled contact analysis with 20-node brick elements the results could be incorrect if contact was on the 9-10-11-12 face.

20 When using linearized contact in a coupled analysis with higher order elements, the results could be incorrect.

Coupled Analysis

1 In a coupled analysis with creep using the CREEP INCREMENT option, the results were incorrect due to a time step conflict between the fixed time stepping procedure in the stress pass and the adaptive time stepping procedure in the thermal pass.

2 In a coupled fluid-thermal transient analysis, the solution was incorrect if full Newton-Raphson is chosen on the CONTROL option.

3 In a coupled fluid-thermal transient analysis the control parameters for the temperature given on the CONTROL option were ignored.

4 In a coupled fluid-thermal-solid analysis, point fluxes were handled incorrectly.

Dynamics

1 Request of Inverse Power Sweep using single eigenvalue method was not working; instead, double eigenvalue extraction was always used.

Element Formulation

1 Membrane element 200 (6-node stress element) only had a linear strain definition. Even if parameter options as LARGE STRAIN or LARGE DISP were used, the strains were still engineering strains.

2 The definition of the local element system for membrane elements 197 (6-node heat transfer element) and 200 (6-node stress element) was not according to the definition given in Volume B.

3 The material orientation defined using the COORD SYSTEM option was not calculated correctly for element 185 (8-node solid shell element).

4 For membrane elements 197 (6-node heat transfer element) and 200 (6-node stress element), the locations of the integration points used to set up the operator matrix and to output the integration point results were not in agreement with the locations given in Volume B.

5 For element 185 (8-node solid shell element), the following options were not supported:
(1) stress-based separation using extrapolated integration point stresses; (2) stress-based friction; (3) stress-dependent contact heat transfer coefficients in a thermal-mechanical coupled analysis.

6 The internal forces calculated for a grounded cbush element were incorrect (1-node 2D cbush element 194 or 3D cbush element 195).

7 In rare cases, an analysis would stop with exit,1005 when using the ORIENTATION option with a direction specified by COORD SYS. If no such exit message occurred, the option was correctly processed.

8 When beam elements are used with a length in the order of magnitude of 1×10^{-5}, the results could be inaccurate.

Fracture Mechanics

1 If the VCCT option was used in the history definition part of the data file to modify an existing crack front using an empty crack front list, the program could crash.

2 VCCT based crack propagation with releasing constraints could in some cases use an existing crack front node as the next node after crack growth.

Heat Transfer Analysis and Thermal Stress

1 The program could crash if a model contains inside-out elements (this applies in general to a non-mechanical analysis).

2 The thickness integration for non-composite heat transfer shell elements with a piece wise quadratic distribution of the temperature over the thickness was not correct.

3 The results of a radiation analysis with a mixture of composite and non-composite shell elements would be incorrect, both for a linear and a quadratic distribution per layer.

Input, Postprocessing and Output

1 The transverse shear output in the layers (post file and output file) was wrong if the TSHEAR option is used. Note that the interlaminar shear output was correct, and so were the stiffness and response of the structure. This is true for both shell elements using the conventional and fast integration procedure through the thickness. For shell elements using the conventional procedure, also the reported strain energy was wrong.

2 The DMIG-OUT option did not function correctly for global non-symmetric matrices. The program could crash or give wrong results.

3 The FORMING LIMIT option gave wrong results for element 185 (8-node solid shell element).

4 The default entry of the frequency to write DMIG output of matrices (third entry of the second data block of the DMIG-OUT model definition or history definition option) was interpreted incorrectly. Instead of writing the output every increment, writing was switched off.

5 When using the PRINT VMASS option with integer-4 versions, the printed mass value was zero if the number of elements is odd.

6 If the GRID FORCE option is used and appears only is the history definition section of the data file, then the program could crash in the following cases: (1) tying forces are present and grid force output is asked on a nodal basis; (2) there is no recalculation of the operator matrix storage after reading the GRID FORCE option and before solving the global set of equations.

7 Energy and work calculations were incorrect for a linear elastic analysis flagged by the ELASTIC parameter option.

8 When using the option DEFINE,EDGEMT,SET, the program could stop with exit,13 if the EXTENDED option is not used.

9 When post code 697 (ply orientation angle) was used without defining a layer number, the program could crash.

10 The output of currents in a Joule heating was confusing. This is not the Ohmic current, but a distributed current. In version 2008 r1, the name of this output quantity has been changed accordingly.

11 When the GRID FORCE option is combined with one of the ADAPTIVE, ADAPT GLOBAL, RESTART, or REZONE options, the program would crash.

12 The reported work done by external forces was not calculated correctly for fast integrated shell elements with thermal expansion.

13 The nonlinear dashpot forces and dynamic reaction forces were inconsistently calculated.

14 The strain energy computation for large strain viscoelasticity (using e.g. MOONEY or OGDEN) was incorrect.

15 When reading a data file using BUCKLE INCREMENT and the number of increments is larger than the number of elements or nodes, the job would stop with exit,13.

16 Input of data which required a non-sorted node list (e.g., when using the SPLINE option) could occasionally result in exit number 13.

17 The ELEM SORT and NODE SORT options did not function correctly when (1) nonconsecutive elements or nodes are used and (2) when adaptive meshing is used to include new elements and nodes. The NODE SORT option was not protected correctly if velocities or accelerations were requested in a static analysis and if temperatures were requested in an analysis without heat transfer.

18 Reading curve or surface pairs would go wrong if non-consecutive element numbering is used.

19 A Marc 2005r3 style post file could be corrupt if elements don’t support all requested post codes.

20 The PRINT VMASS option (1) did not consider point masses entered via the MASSES option and (2) was wrong in a coupled thermal-stress analysis.

21 The output of global stresses was incorrect if the CRACK DATA option is included

Material Behavior

1 The Kumar work hardening law did not function correctly in combination with the LARGE STRAIN parameter option.

2 The test on the valid range of the Poisson’s ratio values for orthotropic materials using new style table input was incorrect.

3 When for orthotropic materials remanence was defined using a B-H or H-B relation, the program could go into an infinite loop or give wrong results.

4 Kinematic or combined hardening with the Hill yield criterion did not function correctly.

5 In a forming process using the Cockroft-Latham damage model or the Oyane model, damage would disappear in areas with compression.

6 The results of an analysis involving curing could be incorrect if recycling occurred.

7 In a fluid or fluid-thermal analysis, the following problems occurred: (1) with new style table input, entering the viscosity as a tabular function of the strain rate caused the program to prematurely terminate with an invalid input error; initial temperatures and volumetric fluxes were handled incorrectly; (2) with old style input, the piece wise linear non-Newtonian viscosity did not support both temperature and strain rate dependence; (3) with both old and new style input, a temperature dependent heat capacitance matrix was not calculated correctly.

8 The thermo-mechanical shape memory model gave incorrect results with the LARGE STRAIN option; the results were correct if PLASTICITY,3 or PLASTICITY,5 was used.

9 The Gurson and Kachonov damage models cannot be combined with F^eF^p plasticity, but there was no automatic switch to additive plasticity.

10 In cases where progressive failure is the only nonlinearity in the model, the operator matrix might not be re-assembled.

11 The soil material model could give wrong results in a large deformation analysis (it was possible to get a negative porosity).

12 The curing material model could give incorrect results if (1) iterations occurred within an increment or (2) cure kinetics model 3 was used.

13 The NLELAST model would yield wrong results in the following cases: (1) model 1 is used and only a tension curve is defined; (2) model 4 is used in a plane strain case and $\sigma_{33} > \sigma_{cutoff}$; (3) reduced integration elements with hourglass control are used.

Parallel Processing

1 When running a DDM analysis using the SPLINE option, the program could crash in the following cases: (1) use was made of combined user input and automatic detection of nodes/edges with a normal vector discontinuity; (2) the analytical representation of a deformable body was written into an mfd-file during increment 0.

2 When using local adaptive meshing, the automatically generated multi-point constraint equations could be incorrect for nodes on the domain boundaries.

3 In rare cases, there could be a program crash in a DDM analysis after having read the restart file and continuing the analysis.

4 Nodes on inter-domain boundaries were not treated correctly in the direct solver if some elements connected to these nodes are deactivated.

5 When using element 200 (6-node membrane element), the solver convergence could be poor.

Restart

1 If a restart file was generated using a control file with the RESTART NOW and STOP NEXT options, then the analysis using this restart file would stop immediately after reading the restart file.

2 The program could crash after the second restart (or later) if a model is using surfaces with trimming curves entered via the ADD RIGID or CHANGE RIGID option.

3 The program could crash upon restart if a model contains the POINTS or SURFACES option.

Solvers

1 When running jobs with user subroutines on Windows XP with Microsoft Visual Studio .NET 2005, the CASI iterative solver (SOLVER 9) could cause a program crash if out-of-core matrix assembly was used.

2 Possible program crash, non-converged solution or wrong results if the incomplete Choleski preconditioner was used for the sparse iterative solver (SOLVER 2) in combination with a nonpositive definite matrix (e.g. due to Mooney material behavior). Note that wrong results appeared only if a non-converged solution was accepted by the user.

3 The maximum number of iterations performed by the CASI iterative solver (SOLVER 9) was not the maximum number that the user requested.

Table Input

1 A welding analysis gave incorrect results when using a disc shaped surface weld flux in combination with new style table input.

2 Modal superposition did not work correctly with new style table input.

3 Initial conditions INIT STRESS, INITIAL PLASTIC STRAIN, INITIAL PORE, INITIAL POROSITY, and INIT PC did not work correctly with new style table input.

4 With new style table input, the internal program flag to reassemble the operator matrix when there is a change in a kinematic boundary condition was set incorrectly for a nonmechanical analysis.

5 With new style table input, a film loading on element 44 (8-node heat transfer brick element) defined by distributed load types 1, 5, 7, 9, 11 or 13 was not accepted by the program (note that this would never happen if the data file was prepared using Marc Mentat, since then the Face ID method is used).

6 With new style table input, general traction loads (distributed load type 21, DIST LOADS option) did not work for tetrahedral elements.

7 When using new style table input, the magnitude of distributed loads was not written on the post file. As of version 2008 r1, the magnitude of all the boundary conditions, corresponding to their first evaluation point, will be written on the post file as global variables.

8 When using multiple load cases with AUTO INCREMENT and new style table input, the applied load could be incorrect (except for the first load case).

9 For nonlinear springs, tables specifying the dashpot force were always extrapolated, regardless of user input.

10 With new style table input, the work done by foundations was not available. In version 2008r1, the foundation work is now made available in the output file as part of energy print out. Similar to the old style input, the calculation is based on the assumption that foundations are linear.

11 With new style table input, for user routines NEWSV, INITSV, NEWPO, INITPO, and INITPL, the quantities to be defined by the user were not initialized to zero in the calling program. Hence, if the user does not explicitly define these quantities within the user routine, wrong values (obtained from a previous call) could be used.

12 With new style table input, the movement of a rigid contact body added using the ADD RIGID option was incorrect.

13 The table ID associated with the QVECT option was not printed.

14 The case that equations are used to define loading and unloading curves for a gasket material was insufficiently protected.

User and Utility Routines

1 Element data obtained via a call to utility routine elmvar was wrong for elements which are not part of the current element group.

2 The convergence behavior for beam elements using the HYPELA2 user subroutine was poor if the material model was based on the mid-increment strain definition.

Marc Mentat

General

1 On Windows, if a model file was opened for which the total length of path and filename exceeds 96, Mentat would crash when a subsequent command was given.

2 The run_python script in the bin directory could fail to work on platforms on which the standalone Python interpreter py_post.so depends on compiler shared libraries. This problem was reported on Linux SuSE 10.

3 It was not possible to use a redirection symbol (> or <) when a SYSTEM COMMAND was given.

4 On Windows, the various scripts in the bin directory could fail to work if Mentat was installed in a directory of which the path contained blanks.

5 Running a procedure file, generated on Microsoft Windows, on a Unix system caused problems with strings that are used as a filename (or any string in which Mentat allows blanks). The carriage-return would become part of the string, which for filenames would cause an error.

6 Defining a node path on a chain of line elements in case no branches exist and only 2 nodes are picked could result in an incorrect path.

7 Invisible symbols were created for data points outside the visible rectangle for all XY-plots (path plot, history plot, tables, etc). This would cause a problem with the FILL VIEW command. Since these symbols where included when the view was computed, the graph could become very small in some cases.

8 The labels to the axes of the generalized XY plotter, and in particular the multiplication factors, would not be updated if the X or Y limits of the plot were changed.

Preprocessing

1 There was no button to switch between GENUINE and LINEARIZED contact between QUADRATIC SEGMENTS in the ADVANCED CONTACT CONTROL menu for non-structural analysis classes allowing contact, such as HEAT TRANSFER or DIFFUSION.

2 Face loads in the V SHEAR direction were drawn incorrectly. The arrow pointed in the opposite direction.

3 The plotting of arrows in SOLID mode for face and edge boundary conditions was incorrect. The arrows seemed inside out.

4 The menus for JOULE EDGE FLUX and JOULE FACE FLUX were incorrect. They now look identical to their thermal counterparts.

5 When expanding TRIA3 elements of type 201, 6 and 2, the type for the resulting PENTA6 elements was not set to 136. Also, when expanding TRIA3 elements of type 37 and 38, the type for the resulting PENTA6 elements was not set to 137.

6 The static menu for MESH GENERATION->EXPAND->ADVANCED EXPAND wasn't being displayed.

7 When graphically selecting NODAL TIES, SERVO LINKS, SPRINGS/DASHPOTS, RBE2s, RBE3s or RRODs, picked entities were not highlighted.

8 All toggle buttons in the SELECT BODIES FOR DMIG OUTPUT popup menu would be displayed as "on" as soon as just one body was selected.

9 For an electromagnetic-thermal job, if a material is missing the electromagnetic material data, the program would crash when writing the input file, submitting a job or performing a job check.

10 On AMD Opteron and EM64T running Linux, submitting a DDM job from Mentat would fail if the Marc version used Intel MPI. This was due to mismatches between python versions on the machine.

11 In the MATERIAL PROPERTIES->...->PLASTICITY menu, the buttons to associate a table with the various parameters for the CHABOCHE method were missing. Note that the tables could be selected by entering the commands via the keyboard.

12 For a load-controlled rigid body, the coordinates of the control node, as can be seen in the CENTER OF ROTATION menu section, were grayed out if the body had no rotational degree(s) of freedom (the auxiliary node had not been set). The coordinates are still meaningful for the output of moments.

13 A problem could occur when working with a model file that was corrupted by an older bug in REMOVE UNUSED MATERIALS. The old code would remove only the unused top materials, and leave all the non-top materials underneath in the model. A model that had been saved after this operation may therefore have had a non-top material as first material.

When reading a model, the current material was set to the first material in the model. In the case of the model described above, the current material would then point to a non-top material. This again would allow the user to add elements to this non-top material, and as a final consequence the program could crash on a job check or when writing the input file.

14 The name of the USHRINKAGE user subroutine was missing in the menus, and USHRINKAGE would not be listed in SELECTED USER SUBS.

15 The RAPID MOTION SPEED setup for NC machining was inconsistent with the Marc definition. A new option, IGNORED, which specifies that the time spent for the rapid cutter motion is to be ignored, has been added and is now the default.

16 The list of loads in the ASSEMBLE LOAD VECTOR menu was incorrect.

17 Nodal tying type 23 was using 2 retained nodes. It has been corrected to 1 retained node.

18 The TRIMMING menu has been removed from loadcases MECHANICAL, COUPLED, and JOULE-MECHANICAL.

19 If LINEAR ELASTIC ANALYSIS option was selected, the FOLLOWER FORCE option was not greyed out.

20 The orientation directions created by the `*orient_local` command were not perpendicular if the element was not rectangular.

Marc Writer

1 Face loads in the V SHEAR direction would be written incorrectly to the Marc input file if new-style (table-driven) input was used. The sign was wrong.

2 An incorrect Marc input file would be created if old-style (non-table-driven) input was used and a STATE VARIABLE or PORE PRESSURE boundary condition or a STRESS, PLASTIC STRAIN, RELATIVE DENSITY, PRECONSOLIDATION PRESSURE, POROSITY, VOID RATIO, STATE VARIABLE, PORE PRESSURE, or DEGREE OF CURE initial condition was applied to elements of type 136. The boundary or initial condition was only applied to the first two integration points.

3 An incorrect new-style (table-driven) input file would be written for a model containing a TEMPERATURE or PRESSURE initial condition, using the POST FILE method in conjunction with the LAST INCREMENT option.

4 If additional rigid contact bodies had been added in a restart job, the set of these bodies would be written to the Marc input file, which would result in an exit 13 from Marc.

5 An incorrect Marc input file would be created if in a Fluid-Solid or Fluid-Thermal-Solid analysis both a mechanical and a fluid point load had been defined, and new-style (table driven) input was used. The "point load" keyword and the subsequent line would be omitted for the fluid point load. A similar problem would occur for distributed loads.

6 For old-style (non-table-driven) input, the NODAL TEMPERATURE boundary condition would be incorrectly written to the POINT TEMP option in the history definition section when using the USINC user subroutine.

7 The middle temperature or middle heat flux for the INITIAL TEMPERATURE initial condition and the FIXED TEMPERATURE and POINT FLUX boundary conditions would not be written in a heat transfer or coupled analysis if a quadratic temperature distribution was selected and the only shell elements in the model were composite shells.

Postprocessing

1 The plotting of CONTOUR BANDS on a 8-noded element face for a nodal quantity was different depending on the EXTRAPOLATION METHOD.

2 The plotting of ISO SURFACES on HEXA and PENTA elements was not correct in for values outside a MANUAL scalar plot range. TETRA elements were ok.

3 The postprocessing of integration point results for the 6-noded pentahedral element types 136 and 137 was incorrect. Only the data for the first two integration points was used.

4 The postprocessing of integration point results for the 6-noded triangular element types with seven integration points (197 and 200) was incorrect when EXTRAPOLATION METHOD:LINEAR was used. The extrapolated values at the midside nodes were wrong.

5 History and path plots were incorrect in case of collapsed elements. The element value at the collapsed node was different from the value shown in scalar plots of the model.

6 Collecting the data for a history plot could be very slow in case of a DDM analysis with multiple post files of a large model involving contact.

7 The deformation magnification factor would be displayed also if the deformed configuration was not shown.

8 In case that the element plot setting was FACES/SURFACE and the post style was CONTOUR BANDS, the automatically calculated range could be incorrect.

9 The name of the post quantity "Electric Current (Integration Point)" has been changed to "Current Density (Integration Point)". Note that old procedure files need to be changed when this quantity is selected in postprocessing.

Marc Reader

1 The convective velocity boundary condition in a convective heat transfer analysis (BOUNDARY CONDITIONS>STATE VARIABLES>CONVECTIVE VELOCITY) was not read from a table driven Marc input file.

2 Face loads in the V SHEAR direction were read incorrectly from a new-style (table-driven) Marc input file. The sign was wrong.

3 The reader could crash if the Marc input file contained a local adaptive criterion on the entire finite element mesh, instead of a selected set of elements.

4 The parameters of the Principal Stress damage model were not translated.

5 Rigid contact bodies, defined as NURBS, that do not have the ANALYTICAL flag would be translated in a discretized form.

6 Import of transformations in 2-D analyses could result in incorrect second and third local directions if the second point of the TRANSFORMATION option did not lie in the global XY-plane. In the extreme case that the second point would lie on the straight line through the first point and the node, the second and third directions were effectively random.

7 Marc input files using the CHABOCHE, KUMAR, POWER LAW, or RATE POWER LAW method for plasticity were not translated correctly.

Material parameters were OK, but table references were missing, and nondefault values for the options for CHABOCHE and KUMAR were not set.

8 Gasket material was translated incorrectly. The base value for the loading and unloading paths was set to 0, which would cause the corresponding tables to be ignored when writing the model to an input file.

9 The cure shrinkage parameter as used in the "table" model for cure shrinkage material data was not translated. Also, the "degree of cure" initial condition was not translated from a new-style (table-driven) input file.

10 A model with multiple SHAPE MEMORY materials would be imported incorrectly.

11 Distributed loads in an electrostatic analysis would be translated incorrectly.

DFX Reader

1 The option to translate a specific LAYER did not work correctly.

Nastran BDF Reader

1 Importing a BDF file, created on a Microsoft Windows system, on a Unix system might produce incorrect data. Ctrl-M characters in the file could cause problems reading the data.

IGES Reader

1 Trimmed surfaces that have periodic NURBS curves were imported incorrectly. Such surfaces and curves are now converted to non-periodic.

2 For a trimmed surface that has a collapsed singular point, the collapsed trimming curve would not be imported, which leads to an open loop of trimming curves. At such a singular point, the coordinates of the control points are identical and the length of the trimming curve is zero in real space.

3 The translator would import both code 308 and 408. It is not necessary to import 308, because 308 (subfigure definition) is the master of 408 (subfigure instance).

STL Writer

1 The output of QUAD4 element faces was not correct. They are now exported as pairs of TRIA3 element faces.

Python

1 The *image_save commands in embedded Python scripts (`py_separate_process off`) would not work. Now the sequence:

```
py_send("*image_save_rgb 1 z1.rgb yes")
py_send("*py_update")
```

will properly save the image file.

List of Known Problems in this Release

Marc Known Problems

Fracture Mechanics

1 The application of distributed load on the faces of a crack that performs VCCT based crack propagation is not supported. The load should be extended to the new crack faces but this is not done correctly. This type of load can be applied together with remeshing if the crack is not growing.

2 The VCCT crack propagation option of releasing glued contact does not work together with remeshing. When crack propagation occurs, the released part of the glued interface is not allowed to glue again. The information about this is lost if the body is remeshed after crack propagation.

3 Parallel processing (DDM) does not work for VCCT if the crack is modeled by connecting two distinct parts with user tyings. In particular, the crack propagation with releasing user tyings is not supported for DDM.

4 VCCT with quarter points in 3-D is not correct.

5 VCCT with higher order tetrahedrals is not available, and is blocked by the program.

Contact

1 The DEACT GLUE option does not work together with remeshing. The information about DEACT GLUE is lost if the body to which the involved nodes belong is remeshed.

2 If contact bodies are present along the cyclic symmetry planes, then ALL elements must be part of a contact body for cyclic symmetry to pick up all the faces. If this is not done then only those elements that are part of a contact body will be handled as cyclically symmetric

3 MOVE and new style tables do not work, turning off new style tables is a work around.

4 Distributed friction should not be used in soil or powder material models, use nodal based friction instead.

Material Models

1 The new curve based material orientation option may give incorrect results for long complex curves. The orientation can be checked during postprocessing by using the element post codes 691 and 694. Workaround: split up the curve into separate smaller curves.

2 Prestate can not be used with damage or progressive failure. There is currently no way to map the damage variable.

3 PLOTV user subroutine does not work with fast integrated composite shell elements.

4 The mixture model type 3 (for nonlinear behavior) is not available with an updated Lagrange analysis.

5 The mixture model type 3 is not available for beam elements.

6 The mixture model type 3 is not available for ADAPT GLOBAL or REZONING if it is used in the body that is being remeshed.

Procedures

1 There is a problem in performing harmonic analysis with an updated Lagrange analysis. The results are not correct.

2 It is not possible to use INITIAL STATE or CHANGE STATE over part of the model and INITIAL TEMP and POINT TEMP over a different part.

Adaptive Meshing

1 Occasionally the global adaptive mesher flips the orientation of the shells, so any distributed load is incorrect

2 Shell remeshing with nodal boundary conditions applied directly to nodes may give the loads at wrong locations after remesh. Workaround: Apply the load on geometric entities instead.

Wear Analysis

1 The wear models defined through RECEDING SURFACE should not be used with shell elements.

2 The body which is subjected to wear should not be remeshed using the ADAPT GLOBAL option.

Table Option

1 The table option does not yet support: Pressure loaded cavities, Fourier, or Element types 31 (pipe elbow) and 51 (cable).

2 HOLD NODE option does not work with table driven input.

Heat Transfer

1 Latent heat is not available with lower- or higher-order wedge elements.

Dropped Capability

1 The previous substructure/superelement capability using NEWDB (parameter), SUBSTRUC (parameter), SUBSTRUCTURE (model definition) and SUPERINPUT (model definition) are no longer supported. The SUPER (parameter) and the BACKTOSUBS (model definition) have a slightly different use. Superelements should now be created using the SUPERELEM option and read into subsequent models using DMIG and K2GG.

Marc Mentat Known Problems

Preprocessing

1 When the command `*draw_beam_axes on` is used to display the local element axes for BUSHING elements, the case ORIENTATION TYPE: ALIGNED WITH COORDINATE SYSTEM is not handled correctly.

2 If global layer IDs are used for composite materials, selecting ALL layers in the JOB RESULTS menu for element quantities will not work. The quantity will be output in global layer IDs 1-N, where N is the maximum number of layer per element.

3 The default stiffness degradation method for progressive failure of composites is now the new “GRADUAL SELECTIVE” method. Existing model files and procedure files in which progressive failure was selected will now use this new method instead of the pre-2007r1 method which is no longer recommended. If desired, the latter can still be used by selecting the “STANDARD” degradation method.

4 If a coordinate system references another system and if the latter changes, the plotter will not update the graphics.

5 No error message is given when the option LOAD ACTIVE IN CONTACT is off and old style (non-table-driven) input is used. Switching this option off only works with new-style input.

6 The collection of nodal data for a history plot can be very slow if a large number of distributed loads is present.

7 If a model contains solid section beam elements with CALCULATED properties or thin-walled section beams with a GENERAL section and ALL LAYERS is selected for an element quantity in JOB RESULTS, an incorrect input file will be written. You must enter a LIST of layers in this case.

8 Plotting solid section beam elements in 3-D solid mode does not work.

Marc Reader

1 Reading of Marc input file history definition is only possible for Marc input files using the new style table input. A warning message will be issued in the Mentat dialog area if an old style Marc input file has been detected, for which the history definition cannot be read.

2 Upon reading an input file containing the INCLUDE option, the contents of the included file are expanded.

3 RECEDING SURFACE data is not read in.

4 MIXTURE data is not read in.

5 CONM1 and CONM2 data is not read in.

Troubleshooting Tips

Marc Troubleshooting

1. **New Style Tables**
   a. New style tables are now the default in Mentat and should a previous Mentat procedure file fail to run this could be the cause.
   b. You are encouraged to switch to the new style table format since by doing so Mentat will read the history definition of the Marc input file.
2. **Contact**
   a. If a previously running problem fails, check if there are hard-wired values for contact parameters (e.g. contact zone tolerance, separation force, etc.). In such cases, the defaults may work better.

Note: Under certain conditions, hardwiring of CONTACT parameters may be necessary to model certain physics but if it is done solely for the purpose of making a job run then one could try switching it to default values.

   b. In case convergence is difficult to achieve, discarding initial stress stiffness (through CONTROL option) matrix in elastomer analysis may help. Similarly, taking only the tensile part of the stiffness in shell analysis involving high compressive stresses also can help (this should not be done for eigenvalue analysis).
   c. Use a bias of equal or greater than 0.95 for contact problems involving rigid-to-deformable contact or frictional contact may help in obtaining better results. This is now a default in Marc Mentat 2005 and beyond.
   d. When a problem does not converge well with friction, it is advisable to first ensure that the problem is running well without friction to rule out model set up problems. For problems with friction, the bilinear friction model generally gives the best performance.
   e. The nodal based friction in general provides better results (except for specific cases where deformation involves large compressive stresses in forging applications). For the structural elements – beams, shells, trusses and membranes, the nodal based friction must be used.
   f. In a 2-D contact analysis, the default limit angle between adjacent segments of a contact body is 8.625 degrees, which may play a role if curved structures are modeled using relatively coarse mesh or patches. If there is a significant amount of sliding such that nodes slide from one segment to another, this angle value may cause the nodes to be temporarily stuck at the intersection of two adjacent segments. Sliding to a next segment takes place after separating from the first, which can result in more iterations (or sometimes even non-convergence) compared to smooth sliding. If this happens, increasing the default value of this angle (e.g. to 20 degrees or higher) may speed up the analysis.

Occasionally, similar problem may happen for 3-D analysis and the angle should then be increased to higher than the default value of 20 degrees.

g. When the default separation force/stress is used in a contact problem and the separation behavior is not as expected, one should carefully review the solution to understand the reason. Since the default separation force is set to the maximum residual force in each iteration, nodes not separating could be because the maximum residuals are rather large in the solution. In this case, either specifying a smaller separation threshold or allowing the residuals to become smaller through a tighter convergence tolerance could help. On the flip side, too many nodes separating due to extremely small residuals could also be avoided by providing a larger separation threshold.

h. When a load controlled rigid body is used in an analysis, it can be specified with one control node (controlling translational motions only, with no rotations allowed), or with two control nodes (one controlling the translational motions and the other controlling the rotational motions). Note that when the load controlled rigid body is in contact with one or more deformable bodies, sufficient constraints (nodal boundary conditions or springs or gluing) should be provided to the system of bodies such that the load controlled body is free from rigid body translations and rotations. Without proper constraints, the analysis will terminate prematurely with exit 2004 due to singular equations. Also note that degrees of freedom for rotational nodes in the User Interface/ input deck should correspond to DOF 1 (in 2-D) and degrees of freedom 1, 2, and 3 (in 3-D).

i. APPROACH, SYNCHRONIZE options must be used cautiously in conjunction with position controlled rigid bodies. When the position of the body is specified by the user and this position is abruptly modified during the APPROACH loadcase, the body could revert back to the position specified by the user after the APPROACH loadcase. The typical work-around is to use velocity controlled bodies.

j. A useful aid for trouble-shooting contact problems is to use PRINT,5 parameter in the input deck (in Marc Mentat, it can be activated by JOBS-> MECHANICAL-> JOB RESULTS-> OUTPUT FILE-> CONTACT). This provides contact related information about nodes touching, nodes separating, nodes moving from one patch to another, etc. in the output file.

3. **Load Stepping**

a. For unstable quasi-static analyses, the load incrementation based on the damping strain rate, as defined using the AUTO STEP option, is recommended. This can be activated by the button LOADCASES-> MECHANICAL-> STATIC-> ADAPTIVE MULTI-CRITERIA (PARAMETERS)-> DAMPING STRAIN RATE under NUMERICAL CRITERIA. Usually, the default damping ratio of 2e-4 should provide an efficient and accurate solution. If needed, additional user-defined criteria can be added to introduce other bounds on the applied load increments.

b. Since temperature boundary conditions in heat transfer or thermally coupled analysis are applied instantaneously, it may be sometimes difficult to satisfy the tolerance for allowable temperature change for adaptive stepping procedures like TRANSIENT and AUTO STEP. This can be solved by either increasing the tolerance for allowable temperature change, or by using a fixed stepping procedure like TRANSIENT NON AUTO to ramp the applied temperature.

c. For dynamics problems using the Newmark-Beta or Single-Step Houbolt operators, AUTO STEP checks on the time integration errors and suitably cuts the time step. For high frequency problems or problems with a lot of numerical noise (for e.g. chattering nodes in contact analysis), these cutbacks could cause the time step to be too small. In this case, the feature for checking on time integration errors can be turned off by setting the 3rd field of the 3rd data block of AUTO STEP option in the input file to 1 or via the button TIME INTEGRATION ERROR CHECK.

d. If the CHANGE STATE option using a thermal post file does not seem to work properly in conjunction with AUTO STEP, make sure that the transient time in the thermal post file matches or is larger than that used for the mechanical analysis.

e. When AUTO STEP procedure is used for adaptive load stepping and the analysis does not seem to be increasing the time step sufficiently even though convergence seems to be okay, the desired number of recycles could be increased from a value of 3 to a higher value, e.g. 5 (this is now a default since MSC.Marc Mentat 2005). This is particularly useful for problems with displacement checking, where a minimum of 2 recycles is already used to establish convergence.

4. **Materials**

a. When tables are used to specify variations in material properties (e.g. Young's modulus, yield stress, etc.) with analysis variables (e.g. temperatures and equivalent plastic strains, engineering strains), the data should be provided over the entire range of analysis variables expected to be encountered in the analysis. Failure to do so can cause the material data to be extrapolated to non-physical values resulting in analysis failures (this is very often seen with elements turning inside out or node incorrectly projected on or sliding off the contact surface message).

b. When the coefficient of thermal expansion is specified as a function of temperature, the instantaneous coefficient of thermal expansion needs to be specified (refer to Chapter 6 of *Marc Volume A: Theory and User Information*).

c. When rapid changes in elastic strains are encountered in an implicit creep analysis due to changes in loading, bending, or other non steady-state conditions, there is a chance that, in conjunction with the secant tangent scheme, the analysis may encounter a nonpositive definite system of equations in cycle 1 of the mechanical pass. This is usually related to the fact that a large inelastic strain increment was predicted by a default steady state creep formulation used in cycle 0 of the increment. This can usually be solved by either of the following workarounds:

- flag a nonpositive-definite solution. This usually allows the solution to proceed without impacting the super linear convergence characteristics of the scheme
- change the flag for the tangent scheme to 3 instead of 1 on the CREEP parameter. This undocumented flag deactivates the steady-state creep predictor in cycle 0. While this avoids the nonpositive definite system, it could impact the convergence characteristics of the solution.

5. **Remeshing**

a. If 3-D tet remeshing fails, check for:

- self contact: this can cause the mesher to fail. This is a current limitation.

- sharp angles in rigid body: the sharp angle can penetrate deformable body in such a great amount that the new mesh's nodes or elements may be created inside the rigid body. Try to avoid sharp angle or use small elements in those areas, say, using the curvature control to place smaller elements in those area.
- very thin section and large penetration: this can also cause mesher failure as projection of new nodes to the contact surfaces becomes difficult.
- deformable-to-deformable contact: try to use different mesh size for each contacting bodies such that the lower numbered contact body has a denser mesh.

b. If there are questionable results:

- then avoid unnecessary remeshing – as remeshing needs to map data from old mesh to new mesh where there is a big change in element size.
- due to data mapping, the results in the remeshing increment may show some discontinuity. This is normal.

c. Selection of appropriate meshers:

- In 2-D remeshing, do not use overlay mesher if there is self contact or if there is a hole inside the deformable body. Use advancing front mesher in such situations.
- Triangular mesher can be useful if the geometry of the deformable body has or will have a sharp corner and cannot be meshed properly by using the quad, or degenerated quadrilateral elements. The tape peeling user guide example shows the capability of using the triangle remeshing. However, appropriate element type must be chosen if the problem has large deformation.

6. **Restart**

a. If a restart analysis does not seem to be applying the applied boundary condition history correctly, you need to make sure that the boundary condition history has been suitably modified to account for the fact that a portion of the analysis has already been completed. There are three ways to accomplish this:

- switch to table driven input procedure
- shift the X-axis of tables in the User Interface and write out the portion that remains to be analyzed
- copy the original input file to a new location, set up the RESTART option and then delete the portion of the analysis that is already completed.

b. If a restart analysis produces Exit 77 though nothing significant seems to have been changed, try inserting the REAUTO option just below the RESTART option and 0,0,1 in the following data option. In Marc Mentat, this can also be flagged by using the IMMEDIATE option under the JOBS-> MECHANICAL-> JOB PARAMETERS-> RESTART-> COMPLETION OF UNFINISHED LOADCASE menu.

7. **Memory Issues in Large Problems**

a. To run large problems that address over 2 GB of memory, you will need a 64-bit operating system. All versions for the 64-bit systems allow the allocation of over 2 GB of memory.

These versions come in two variants for most platforms. You can (here called the i4 version) use standard Fortran integers as internal pointers. The other (i8 or true 64 bit version) uses long integers (integer*8) for all integers used in the program. The i4 version has a limitation of 8 GB for each memory section. For example, the general memory part itself can use a maximum of 8 GB and solver 8 additionally 8 GB. For a parallel job, this limitation is for each domain. The i8 version has no such limitation.

In order to maximize the size of a model to run, try the following steps:

- Run in parallel on a cluster. This will utilize the memory on each machine of the cluster.
- On a multiprocessor machine using the i4 version where the available memory on the machine is more than 8 GB, decompose the model such that each domain uses less than 8 GB for each memory part.
- Use an iterative solver (solver 2 or 9). This would help tremendously both in terms of memory and speed. The CASI solver (solver 9) is better than solver 2 in handling ill-conditioned systems, for example models with shell or beam elements. For solver 2, the diagonal preconditioner uses less memory than the incomplete Cholesky preconditioner. For solver 9, the OOC,,1 parameter has the effect that some parts of the solver use disk storage. This save memory with a small penalty in performance.
- Use the ELSTO option in the parameters section. This writes element quantities to disk.
- Use the out-of-core solver for the default solver, solver 8. This will be done automatically for a nonparallel run if the solver memory would go beyond the physical amount of memory available on the machine. It can also be imposed using the OOC,,1 parameter

In addition, even though the hardware may have adequate memory, the operating system may have to be re configured to allow large amounts of memory to be used. On some systems, user limits also may have to be adjusted (e.g. ulimit command on many systems will allow larger data size). Please check with your systems personnel if unfamiliar or unable to check the above.

b. To pre- or post process more than 2 Gbyte of data in Marc Mentat, the 64-bit version needs to be used on a 64-bit operating system. If the 32-bit version of Marc Mentat is used, the post file can be larger than 2 GB on systems where large file support is available; however, the amount of data in each post-increment must be less than 2 GB.

c. The ALLOCATE parameter can be used to specify how much memory Marc should allocate initially for general memory. If it is too small or not set, Marc will automatically reallocate memory as needed. For most problems, you will not have to adjust the option, simply let Marc reallocate memory.

   However, for large problems, the reallocation process can be time consuming and may fail to get memory if the process tries to get a block of memory but goes out of system limits (note that most systems do not temporarily release the previously allocated memory block till the reallocation process is completed). In such cases, it is more efficient to allocate a large block of memory initially, and let Marc fill it up as your job progresses.

d. For serial jobs using the multi-frontal solver (solver 8), it may be advantageous to allocate memory for the solver workspace initially. This can be performed using the PREALLOC parameter. This may be done using the JOB-> ...-> JOB PARAMETERS-> SOLVER menu

in Mentat. The advantage of this is while memory may be wasted, there will be sufficient memory for the decomposition phase. Also, it can avoid costly reallocations of memory similar to the case with general memory.

8. **User subroutines**

   If there are problems in jobs with user subroutines, a variety of approaches are available for troubleshooting:

   a. Debug and fine-tune a user subroutine on a small test model before applying it to the actual finite element problem.

   b. Run the user subroutine as a stand-alone program, provide a wide range of inputs to the program and make sure that the outputs are stable numbers.

   c. If division expressions are being used, make sure that the denominator cannot go to zero. Extra precaution may be needed for increment 0 or 1, where many quantities are initialized.

9. **OpenGL**

   If you run into display problems when running the OpenGL version of Marc Mentat, you may want to try the following options:

   a. mentat *-glflush*

   b. mentat *-ss* off

   Symptoms could be either the graphics will not be updated regularly, or you experience sluggishness when selecting nodes or elements.

List of Build and Supported Platforms

Marc Platforms

Vendor	OS	Hardware	FORTRAN Version	C Version	Default MPI	Also Works On
HP-Alpha (DEC)[4] HP (64-bit)[2, 4] HP (64-bit)[2, 4]	Tru64 5.1 HPUX 11.11 HPUX 11.23	Alpha Server 4100 PA2.0 Itanium 2	f90 5.5 f90 3.1 f90 2.8.7	cc 6.4 A.03.73 A.06.02	HP MPI 2.0 HP MPI 2.0 HP MPI 2.2	
IBM (64-bit)[4]	AIX 5.2	RS/6000 & RS/6000 SP	xlf 8.1.1	cc 6.0.0	MPICH[1]	AIX 5.3
SGI (mips4 64-bit)[2, 3, 4] SGI (Altix 64-bit)[2, 4]	IRIX 6.5 Linux 2.4.21 - sgi303r2	R12000 Itanium 2 (Propack 3.0)	f90 7.4 Intel 9.1	cc 7.4 Intel 9.1	MPICH[1] SGI MPT 1.13	 Propack 4.0
Sun (64-bit)[4] Sun (64-bit)[4]	Solaris 10 Solaris 10	UltraSparc III x86	f90 8.3 f90 8.3	cc 5.9 cc 5.9	MPICH[1] SUN HPC 7.1	
Linux (32-bit) Linux (64 bit)[4] Linux (64-bit) Linux (64-bit)[4]	RedHat AS 4.0 RedHat AS 3.0 RedHat WS 4.0 RedHat WS 4.0	Intel Pentium III or equiv. Itanium 2 AMD Opteron Intel EM64T	Intel 9.1 Intel 10.1 Intel 9.1 Intel 9.1	Intel 9.1 Intel 10.1 Intel 9.1 Intel 9.1	HP MPI 2.2.5.1[5] HP MPI 2.2.5.1[5] Intel MPI 3.1[6] Intel MPI 3.1[6]	 RedHat AS 4 SuSE 10, Intel 10.1 AMD Opteron, SuSE 10, RedHat 5, Intel 10.1
Intel (32-bit) Intel (64-bit)[4, 8]	Windows XP SP2 Windows Server 2003 x64	Intel Pentium III or equiv. Intel EM64T	Intel 9.1 Intel 9.1	Intel 9.1 Intel 9.1	Intel MPI 3.1 Intel MPI 3.1[7]	Intel 10.1 Vista XP 64

[1] Hardware MPI version also available (via *maintain in /tools* directory).
[2] Supports Solver 6.
[3] Supports multi-threading.
[4] Supports true 64-bit version.
[5] Supports the Intel MPI 3.1.
[6] Supports the HP MPI 2.2.5.1.
[7] Supports the Microsoft MPI 1.0 (SP1).
[8] The LP64 (i4) version supports only serial runs.
Parallel is enabled for all platforms.

Marc Mentat Platforms

Vendor	OS	Hardware	Also Works On	ACIS2
HP (64-bit)	HPUX 11.0	PA2.0		R17
HP (64-bit)	HPUX 11.22	Itanium 2	HPUX 11.23	R17
IBM (64-bit)	AIX 5.2	RS/6000	AIX 5.3	R17
SGI (Altix 64-bit)	Linux 2.4.21 sgi303r2	Intel Itanium 2 (Propack 3.0)	ProPack 4.0	-
Sun (64-bit)	Solaris 2.9	UltraSPARC III	Solaris 2.10	R17
Intel/AMD (32-bit)	Linux (32-bit)	Intel Pentium III or equiv.		R17
Intel/AMD (64-bit)	Linux (64-bit)	Intel EM64T or AMD Opteron		R17
Intel/AMD (64-bit)	Linux (64-bit)	Itanium 2		-
Intel/AMD (32-bit)	Windows XP	Intel Pentium III or equiv.	Vista	R17
Intel/AMD (64-bit)	Windows Server 2003	Intel EM64T or AMD Opteron	XP 64-bit	R17

[1] On SGI Altix and Itanium 2 Linux, the 32-bit IGES, DXF and VDA interfaces for Linux Intel Pentium are used.

[2] To save model files containing solids on ACIS R17 platforms such that they can be read on ACIS 13sp5 platforms, execute the command *acis_version_13 (FILES>INTERFACES EXPORT>ACIS OPTIONS>VERSION 13.0) prior to saving the model file. This command should also be used prior saving model files containing solids in Mentat 2005 or Mentat 2005r3 format.

[3] Uses Python 2.2. All other platforms use Python 2.5.

OpenGL Compatibility

When running over a network, the following combinations of client machine (where Marc Mentat is running) and graphical server (where the user is viewing the program) have been found to work properly using OpenGL:

Client	Server						
	Compaq	HP	IBM	SGI	Sun	Windows[1, 2]	Linux[3]
HP/Compaq/DEC	y[4]	n	y	n	y	y	n
HP	y	y	y	y[5]	y	y	y
IBM	y	y	y	y	y	y	n
SGI	y	y	y	y	y	y	n
Sun	y	y	y	y	y	y	n
Microsoft Windows	n	n	n	n	n	y	n
Linux	y	y	y	n	y	y	y

[1]Requires additional software (see http://www.hummingbird.com or other vendor of X server software).

[2]The following OpenGL graphics cards have been found not to work:

Compaq PowerStorm 300 and 4D10T
STB Velocity 4400
Intense 3D Pro 3410

[3]Requires MesaGL v3.4 or higher.

[4]Double buffering not available (4.0D version will not work on 5.x and vice versa).

[5]Some buffering problems may occur when changing workspaces.

List of Dropped Platforms

Dropped Platforms

The following platforms compiler, and OS have been dropped as of this 2008 r1 release:

- 32-bit Linux RedHat 9
- HP DEC Alpha
- Microsoft Windows 2000

The following platforms and compilers will be dropped in the next release:

- SGI IRIX
- RHEL 3.x (to be replaced by RHEL 4.x or higher)

Important Notes

Marc Notes

1. The startup script for Marc (`run_marc` or `run_marc.bat`) runs the job in the directory where the command is issued, even if a path to the input file is provided.

   Example:

   ```
   run_marc -j ../otherdir/job
   ```

   The job runs in the current directory. All results files are created in the current directory. No files are created in `../otherdir`; only the input file is read from there.

   Filename extensions are now allowed in the command line options.

   Example:

   ```
   run_marc -j job.dat -u usersub.f
   ```

   A new option: `-dir` directory allows a different working directory to be specified. All created files, scratch files, and results files except the log file and status file are created in the directory specified with this option. This option is not supported through Marc Mentat.

2. When running any of the examples in the *Marc User's Guide* or *Marc Introductory Course*, it is best to copy all the files (`.proc`, `.mfd`, `.mud`, `.t16`, `.t19`, etc.) in the example directory to the current, local directory. This is especially required for the examples where the procedure file uses the previously generated results file or model file to demonstrate the example.

3. Hardware Vendor Provided Solver

   The hardware vendor provided solvers (Solver 6) are available for parallel matrix solution. In a parallel run using Domain Decomposition, this is utilized automatically. This feature can also be used in a serial run in which case only the matrix solution will be performed in parallel. There are two ways to activate this feature:

   a. Using the command line option `-nthreads`.

      Example:

      ```
      run_marc -v no -j test -nthreads 4
      ```

runs the job `test.dat` using four processors for the matrix solution. This is not available from within Marc Mentat.

b. Using the environmental variable MARC_NUMBER_OF_THREADS. This variable is set to the number of processors to be used. Note that it needs to be defined in the same window as the one in which the job is started. If the job is started from within Marc Mentat, the variable needs to be set before Marc Mentat is started. If this variable is set and the `-nthreads` option is used, the value given by `-nthreads` will be used.

4. The parallel version of Marc is delivered with MPICH (public domain MPI) or HP-MPI for most Unix platforms. This version can be used for both single multiprocessor machines as well as for separate machines connected over a network. When running a job over the network a so-called host file should be used, see *Installation and User Notes* for Network version from the *Marc and Marc Mentat Installation and Operations Guide*.

   Note: The host file should not be used in a run on a single multiprocessor machine.

   On most of the platforms using MPICH, it is possible to switch to hardware vendor MPI. Only analyses on single multiprocessor machines are supported in the case of versions using hardware vendor provided MPI. An exception to this are ports with HP-MPI which fully supports the network parallel analysis.

   Note: The Linux RedHat9, SuSE 9, and RedHat AS 3 versions support ScaMPI.

5. Installation related:

   a. If you get an error message of `f90 not found` when running a job with a user subroutine and you know there is a FORTRAN compiler on the machine, its path needs to be provided. A typical example would be the Sun platform where the f90 compiler may live in the `/opt/SUNWspro/bin` directory. This path must be added if you get the `f90` error message.

   b. On a rare occasion, a job can fail to run on certain platforms with a message; for example, on DEC machines `libUfor.so not found` or on Sun machines `libsunmath.so.1 not found`. These files with extensions of `.so` are shared objects and the error message suggests that either the run time libraries are missing from the system or installed in a nonstandard place. This problem can be fixed with one of the following procedures:

      Try relinking the version first by executing the `make_marc` script in the `marc2008r1/tools` directory and run the job with and without user subroutines.

      If the problem persists, check if the `.so` file exists in the `marc2008r1/lib/lib_shared` directory. If it does exist, uncomment the following two lines in the `run_marc` script under `marc2008r1/tools` directory:

      ```
      LD_LIBRARY_PATH = $DIR/../lib/lib_shared:$LD_LIBRARY_PATH
      export LD_LIBRARY_PATH
      ```

      If the first line already exists and points to some other directory, replace it with the new line. Run the job with and without user subroutines once again.

      If the `.so` files do not exist in the `marc2008r1/lib/lib_shared` directory or if the `lib_shared` directory does not exist, contact your system administrator to off load the necessary run time libraries from the system CD.

6. When using the `-host` command line option to run a Marc job, the output will automatically be written to the directories specified in the hostfile. For instance, when running a 4 domain Marc job as follows:

   ```
   run_marc -jid jobid -host hostfile -nprocd 4
   ```

   the output will be written for each domain to the directories as specified in the hostfile. By default, Marc Mentat always will write the hostfile to contain the directory specifications.

   However, the following exception applies to the default described above. On Unix systems using the IBM cluster product POE or the Sun cluster product HPC, the `-host` command line option should never be used. Instead, the `-dir` command line option can be used to customize the location of the output. The *user notes* can be consulted for further information on how to use the `-dir` option.

7. The PLDUMP utility routines are explained in *Marc Volume D*, Chapter 9: Special Routines.

   There are three subdirectores in the `marc2008r1/pldump` directory:

Rev1	stores the routines for building the PLDUMP program under `marc2008r1/bin/pldump`. To rebuild, go to the `marc2008r1/tools` directory and type `make_pldump`. A Fortran compiler is needed to perform any rebuilds.
Rev13	stores the routines for building the PLDUMP13 program under `marc2008r1/bin/pldump13`. To rebuild, go to the `marc2008r1/tools` directory and type `make_pldump pldump13`.
Rev9	sotres the routines for building the PLDUMP2000 program under `marc2008r1/bin/pldump2000`. To rebuild, type `make_pldump pldump2000` under the `marc2008r1/tools` directory.

   PLDUMP13 and PLDUMP2000 do not build on the true 64-bit (i*8) version of SGI IRIX in the Marc 2008r1 release. The existing PLDUMP13 and PLDUMP2000 programs in the true 64-bit version are copies of the regular (i*4) version. If rebuilding is needed, please use the regular (i*4) version.

Platform Specific Notes

Various Machine Notes

1. **SGI Machines:**

   a. When running the parallel version with (hardware vendor provided MPI) the Arrays 3.2 version, the job may not run if the marc installation or run directory path name is very long. Normally, path names up to 256 characters are allowed in Marc, but there is a problem with the Arrays 3.2 version.

   Remedy: install patch 3532 from SGI or upgrade to Arrays 3.x or later.

   b. The current version of Marc requires that the blas library is installed for user subroutines to work. This library is normally installed together with the compiler, but on some versions of IRIX they did not get installed automatically. This is the case for IRIX 6.5.2, 6.5.3, and 6.5.4.

For these versions, the fortran compiler runtime libraries in

```
ftn_eoe.sw and ftn_eoe.sw64
```

need to be installed.

c. The user memory limit must be checked before running jobs that require a large amount of memory. This can be done with the use of the `limit` command. The value for memoryuse should be large enough for the amount of memory needed in the job. To increase this limit, you either have to rebuild the kernel or perform the following steps (requires superuser privilege):

```
su
unlimit -h memoryuse
unlimit memoryuse
su - <your username>
```

This will remove the `memoryuse` limits.

2. **HP Machines:**

a. The HP-UX based Itanium 2 version supports the following interconnect/protocol clusters for parallel processing over networks.

– 10/100 Base-T with IP

– Gigabit (GigE) with IP

– HF (Hyperfabric) with IP

– HF with (Hyper Messaging Protocol) HMP

All Itanium 2 machines come with 10/100 Base-T and gigabit (GigE) Network Interface Cards.

HF (Hyperfabric) is an optional device which is HP's implementation of Myrinet.

HMP is a light weight protocol available over *Hyperfabric hardware*.

It will appear as the following software product if installed:
HyprFabrc-00 B.11.22.00.06 PCI HyperFabric; Supptd HW=A6092A/A6386A

This application is HMP enabled, thereby allowing runtime determination of whether to use IP or HMP when HF hardware and supporting software are installed.

To this effect, the `MPI_HMP` environment variable should be set to 'on' for each line of the appfile.

The protocol used is chosen based on the network cards associated with the hostnames or IP addresses specified in the appfile.

The `MPI_LOCALIP` variable may be used to instruct mpirun to use a specific address assigned by `/etc/ifconfig` to the desired NIC in case it is different from the one returned by `nslookup`.

Performance advantage between IP and HMP protocols on HF interconnect depends on message traffic direction of streaming, and message size.

IP performs better than HMP when message traffic is streaming in one direction, and the message size is small.

b. The HP 64-bit versions can only be run on 64-bit enabled HP-UX. This can be checked with the use of the command:

```
/usr/bin/getconf KERNEL_BITS
```

If the returned value is 64 then the system is 64-bit enabled.

c. Large file support (files > 2 GB) are enabled via the `/etc/fstab` file. The option largefiles must be added to the file-system entry for each device that will need to support large files.

3. **IBM Machines:**

The OpenGL version of Marc Mentat may not operate properly on some IBM platforms graphics adapters. The type of graphics adapter may be displayed using the `lsdisp` command.

4. **64-bit IBM Machines:**

Go to `/etc/security` on the system and change the value in the limits file to the following.

Default:

fsize = -1
core = -1
cpu = -1
data =-1
rss = -1
stack = -1
nofiles = 2000

After changing this, please check again `ulimit -a`. It should return unlimited in all the fields. If not, you may have to reboot the system.

Once you change the `ulimits`, you should be able to write a 2GB file on your standard journal file system.

To write 7 GB file, you have to create another file system with large file enabled journal file system.

5. **Windows 2003 Server and Windows XP Machines:**

To enable addressing beyond the 2 GB limit and up to the 3 GB limit on 32-bit Windows 2003 Server and Windows XP machines, you will need to use the `/3GB` switch in the *boot.ini* file, and relink Marc using the `/LARGEADDRESSAWARE` switch.

This switch should be added to the link line in tools/include.bat as:

```
SET LOAD=LINK /nologo %LINK_OPT% /LARGEADDRESSAWARE
```

6. **Linux SuSE 9, and RedHat AS 3.0:**

The HPMPI version (HP-MPI 2.1) support single multiprocessor machines similar to ScaMPI and the following systems/interconnects:

– IA-32/Myrinet, GiGE, TCP/IP, Infiniband
– Itanium 2/Elan4, Elan3, GiGE, TCP/IP, Infiniband
– Opteron/Myrinet, GiGE, TCP/IP, Infiniband

Selection of the above systems/interconnects is through the following `mpirun` flags: (see `mpirun --help`) `-elan -ELAN -gm -GM -vapi-VAPI -udapl -UDAPL -itapi -ITAPI -TCP`

`mpirun -prot` prints the communication protocol between each host (i.e., TCP/IP, IB, HMP, or shared memory).

The korn shell (*ksh*) must be installed for Marc to operate correctly. *ksh* is not installed by default on either RedHat or SuSE Linux. On these systems, the korn shell package, *pdksh*, must be selected at installation or installed manually after installation.

7. **For Builds using the HPMPI as Default:**

   It may be necessary to set the following environment variable when using the debugger, etc. with the HPMPI build.

   ```
   setenv LD_LIBRARY_PATH DIR/hpmpi/lib/architecture
   ```

   where `DIR` is the location of the *marc2008r1* directory, e.g. `/d1/marc2008r1` and `architecture` is given by the following table.

Vendor/OS	Architecture
IA32 (RHEL 3)	linux_ia32
AMD Opteron (SuSE 9)	linux_amd64
Itanium 2 (RedHat AS 3.0)	linux_ia64
EM64T (RedHat WS 3.0)	linux_amd64

Security

Security Notes

The 2008 r1 release requires the FlexLM 10.8 version to work and stores the license manager (lmgrd) and the vendor daemon in a directory named MSC.Licensing\10.8 for Windows XP/2003 and for Unix platforms, it is `flexlm/<platform>`, where `<platform>` is `aix`, `alpha (Tru64)`, `hpux`, `irix`, `linux`, `solaris`, or `sun`. The default location for the license file is `flexlm/licenses`.

LAPI security has been implemented in the products. The capabilities that require a license are given below with feature names as required in the license file.

1. MARC — license required to run one single processor job or one instance of a multiple processor (parallel) job.
2. MARC_Parallel — license required per processor in a parallel run (for example, a four processor job requires one MARC token and four MARC_Parallel tokens).
3. MARC_Mesh2D — license required for each run requiring automatic 2-D remeshing feature in Marc.
4. MARC_Mesh3D — license required for each run requiring automatic 3-D remeshing feature in Marc.

5.	MARC_ShapeMemory	license required for each run using shape memory model.
6.	MARC_MetalCutting	license required for each run modeling metal cutting operation.
7.	MARC_CrossSection	license required for each run using the Cross Section option.
8.	MARC_Electrical	license required for Joule-Mechanical, Coupled Electrostatic-Structural, and Piezoelectricity.
9.	Mentat	license required for each instance of Marc Mentat.
10.	MARC_Hexmesh	license required for each instance of Hexahedral mesher.
11.	Mentat_ACIS	license required for each instance of ACIS when working (import/export) with ACIS based models.
12.	Mentat_ITI_Access	license required for each instance of, or exporting a file using the DXF, IGES, or VDAFS translators.
13.	Mentat_CMOLD	license required for each instance of CMOLD when working (import/export) with CMOLD based models.

MSC does not support the nodelock license for the Windows Server 2003 x64 edition EM64T port in this release.