# MSC.Marc® and MSC.Marc® Mentat®

Release Guide

Version 2005

MSC SOFTWARE®
SIMULATING REALITY

### Corporate

MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707
Telephone: (714) 540-8900
Fax: (714) 784-4056

### Europe

MSC.Software GmbH
Am Moosfeld
81829 München, GERMANY
Telephone: (49) (89) 431 987 0
Fax: (49) (89) 436 1716

### 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



Part Number: MA*V2005*Z*Z*Z*DC-REL

# C O N T E N T S

**MSC.Marc and MSC.Marc Mentat Release Guide**

# MSC.Marc and MSC. Marc Mentat Release Guide

The release of MSC.Marc 2005 family of products broadly encompasses the following objectives:

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

# I. List of the New Functionalities

There are significant enhancements in various key technology areas in addition to improvements in existing functionality in the MSC.Marc family of products. The extent of the improvements is substantial, spanning a range of industries. A list of new features for both the solver and graphical user interface is given below. For example, Chapter 11 of MSC.Marc Volume E: Demonstration Problems contains several of the finite element benchmarks recommended by the National Agency for Finite Element Methods and Standards (NAFEMS). All of the details can be found in the subsequent sections for the enhancements and modifications in both the MSC.Marc and MSC.Marc Mentat 2005 versions.

## MSC.Marc 2005

## MSC.Marc Mentat 2005

# II. Description of the New Functionalities

## MSC.Marc 2005

## 1. Contact Analysis

The contact capability in MSC.Marc has been significantly modified to improve the accuracy of the solution, increase the robustness, and reduce the computational costs. The contact capability has been expanded to be applicable to coupled electrostatic-structural and diffusion simulations. Furthermore, you can use the table driven input procedure in conjunction with the contact option to increase the flexibility without the need to write user subroutines.

This section highlights some of these enhancements. For more details, select a topic through the help table.

### A. Transformation Enhancements

The TRANSFORMATION option in MSC.Marc (to define local coordinate systems) has been completely rewritten. This lets you specify the coordinates and the degrees of freedom in the same way as MSC.Nastran (cord1c, cord1r, cord1s, cordrc, cord2r, cord2s). It also now supports local coordinate systems for nodes coming into contact and for nodes involved in the INSERT option.

#### MSC.Marc Input

TRANSFORMATION model definition option has been enhanced to accommodate the transformation related changes.

| Help | |
|---|---|
| User's Guide | Thermo-Mechanical Analysis of Cylinder Head Joint with Quadratic Contact |

### B. Tying (Multi Point Constraint) Enhancements

In the previous versions of MSC.Marc, the order in which the ties were applied was fixed and determined by the order in which they were given in the input file. MSC.Marc internally uses ties for several options including TYING, SERVO LINK, INSERT, CONTACT, RBE2, RBE3. For MSC.Marc 2005 using MPC-CHECK in the parameter section of the input file, you can apply the multi point constraints in a correct order by forcing an automatic renumbering of all tying equations.

This makes the combination of the above options much more powerful. For example, you can now use the load controlled body node as a tied node in a servo link.

Additionally, you can write the tying forces and moments to the post file as a nodal vector.

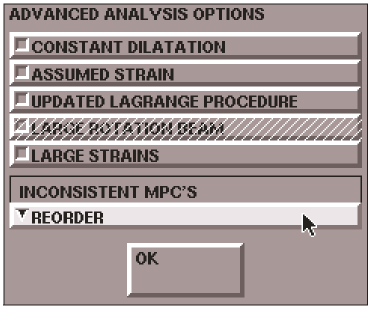


**Figure 1: Advanced Analysis Options Submenu**

### MSC.Marc Input

The tying enhancements are currently activated using MPC-CHECK in the parameter section of the MSC.Marc input file.

| Help | |
|---|---|
| Volume C | MPC-CHECK |

## C. Improvements in Separation Behavior

In some contact problems, within an increment the phenomenon of repeated touching and separating of a node can occur. To avoid an excessive number of iterations, in MSC.Marc version 2003, this repeated touching and separating was allowed only five times per increment per node after which such a node was kept in contact, irrespective of the final contact normal force. Consequently, contact forces could be locally wrong. To solve this kind of problems in MSC.Marc 2005, improved logic has been implemented for nodes which may repeatedly come into contact during the Newton-Raphson recycle process within an increment. Due to this modification, you may observe a very small amount of penetration at the end of an increment. This improved behavior is only available in combination with the iterative penetration checking procedure for which it will be automatically activated.

## D. Deformable Contact

Although there are various options to influence the way in which the contact constraint equations are set up for deformable contact (CONTACT NODE, CONTACT TABLE, and EXCLUDE), the final set of constraint equations may still not be optimal. This may happen, for example, in cases of self-contact or remeshing. In MSC.Marc 2005, an algorithm has been implemented to automatically optimize the set of contact constraint equations, based on the average stiffness of contact bodies, the element edge lengths in the area of contact, and the occurrence of sharp corners. You can activate this method by using the CONTACT option. Contact priorities defined via the CONTACT TABLE take precedence over this option.

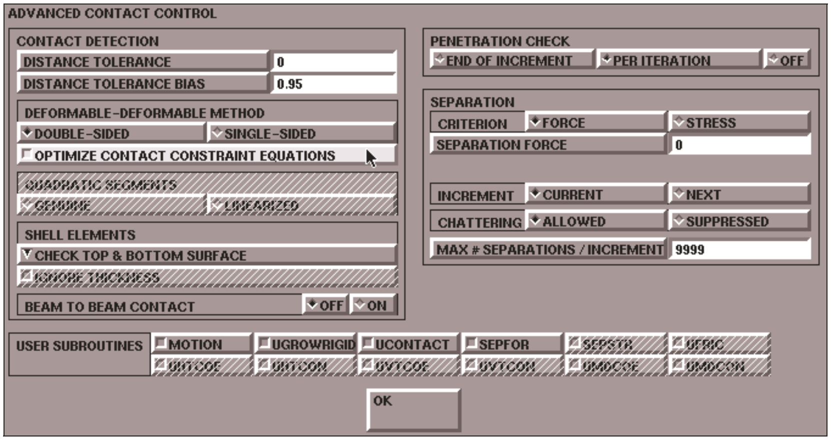


**Figure 2: Advanced Contact Control Submenu**

### MSC.Marc Input

The automatic optimization of the contact constraint equations is activated using the CONTACT model definition option.

| **Help** | |
|---|---|
| User's Guide | Example of optimized contact |
| Volume A | Numbering of Contact Bodies |
| Volume C | CONTACT |
| Volume E | 8.96 |

## E. True Quadratic Contact

The contact normal and contact friction stress when using true quadratic contact are now available for postprocessing both the *contacting* nodes and the nodes of a *contacted* segment.

| **Help** | |
|---|---|
| User's Guide | Thermo-Mechanical Analysis of Cylinder Head Joint with Quadratic Contact<br>3-D Contact and Friction Analysis using Quadratic Elements |
| Volume A | True Quadratic Contact |
| Volume C | CONTACT |
| Volume E | 8.97 |

## F. Parallel Processing

The treatment of contact and user tyings for parallel jobs has been changed to be consistent with the serial jobs. This was an occasional source of different answers in between serial and parallel jobs for deformable-to-deformable or self contact. Also, case of multibody contact and user tyings in the contact area are now treated more consistently between serial and parallel jobs.

## G. Friction Modeling Improvements

In the previous versions of MSC.Marc, the Coulomb friction could be taken into account using one of the two models below:

(1) Arctangent Function Model

The smoothing function is based on the arctangent function which takes into account relative sliding velocity between contacting bodies at the point of contact. Depending on the parameter involved in the smoothing function, the theoretical stick-slip function is approximated with more or less accuracy. Although the global convergence behavior using this model is often satisfactory, it may be difficult to find the optimal value of control parameter, especially for sticking conditions or strongly varying relative velocities during the analysis.

(2) Stick-Slip Model

The stick-slip model is based on the introduction of a sticking or slipping status for each node in contact. Corresponding to a status, different constraints are applied and the iterative solution obtained is continuously checked for correctness of the friction status. The final results obtained in this way are usually accurate, but computational expensive. A drawback of this model is that it cannot be directly applied to 3-D quadratic elements.

Bilinear Function Model

A new friction model is introduced in MSC.Marc 2005 which is more accurate than the model using the velocity-based smoothing function and less expensive and more general than the stick-slip model.

A bilinear, relative displacement-based approximation of the theoretical step function is used (Figure 3). The slope is determined by the slip threshold, $\delta$, that can be automatically determined by the program, but can also be user-defined. Hence, it is not necessary to provide other parameters.

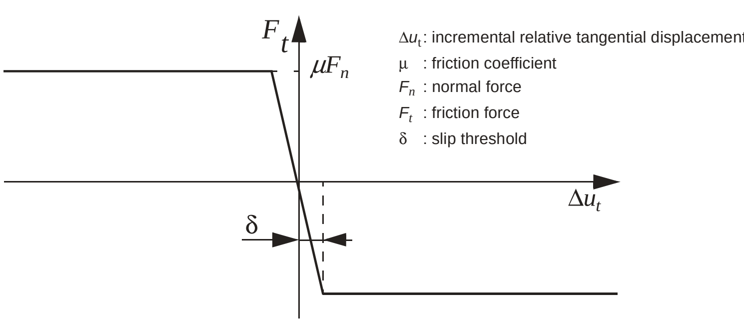


**Figure 3: New Bilinear Friction Model**

The new friction related MSC.Marc Mentat menu is shown in Figure 4, together with the numerical preferences menu, where the initial friction stiffness can be set. Notice that the choice between nodal stress and nodal force based friction does not apply to the new model, where the friction calculation is always nodal force based. You can used this model with either lower-order or higher-order elements.

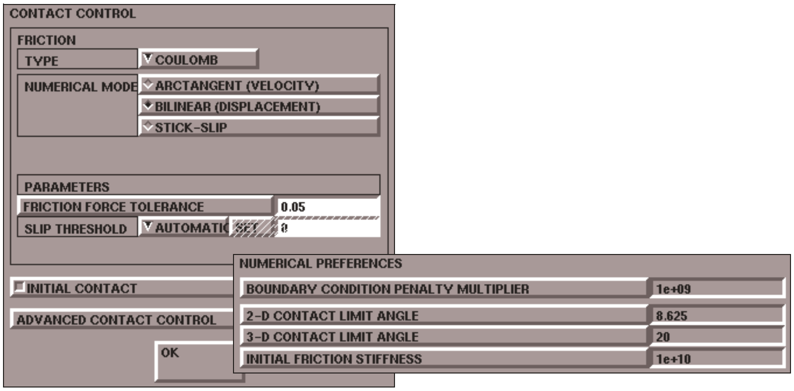


**Figure 4: Friction related MSC.Marc Mentat Menus**

Unlike the stick-slip friction model, the new bilinear model can also be used for metal forming type applications, where yielding of the material introduces a limit on the friction stresses. This reduction of the maximum friction stress is based on the assumption that there is a direct relation between the tangential force and stress at a node. This behavior is activated if the shear friction model is selected.

### MSC.Marc Input

The CONTACT Model Definition option has been modified to activate these new friction models.

| **Help** | |
|---|---|
| User's Guide | 3-D Contact and Friction Analysis using Quadratic Elements<br>Pin to Seal Contact with Various Friction Models |
| Volume A | Friction Modeling |
| Volume C | CONTACT |
| Volume E | 8.97 |

## H. Contact Analysis in Electrostatic-Structural Simulations

The contact capability has been expanded to support electrostatic-structural simulations, one typically represent the electric components in a contact body and the air in a separate contact body. This permits the meshes to be noncongruent at the interface, which simplifies the meshing, and permits the air to be remeshed using the global adaptive meshing technique, when large motion of the devices occur.

## I. Contact Analysis for Diffusion Simulations

The contact feature has been expanded to be used in conjunction with diffusion analyses. This may be used to analyze regions in contact, but with dissimilar meshes or for the interaction between a body and the environment.

## J. Radiation to the Environment

The thermal contact capability has been enhanced such that if a face is not in contact with another face, it may have radiation to the environment in addition to convection to the environment.

| Help | |
|---|---|
| Volume A | Contact radiation |

## K. Use of Tables

You can now use tables to define many of the contact parameters such as the coefficient of friction or the thermal coefficients between bodies or to the environment. As an example, in a coupled thermal mechanical analysis, you can enter a table or an equation to have the coefficient of friction be a function of the normal stress and the temperature without using a user subroutine.

| Help | |
|---|---|
| User's Guide | 3-D Contact and Friction Analysis using Quadratic Elements |
| Volume C | CONTACT with TABLES; CONTACT TABLE with TABLES |
| Volume E | 8.13 |

## L. Adding and Changing Rigid Surfaces

In multi-stage manufacturing simulations, it is often desirable to perform part of the simulation and then change the geometry of the rigid tools or add new tooling. This is now possible in this release of MSC.Marc. To perform this type of analysis, you must restart the job and indicate which contact bodies are changed or added on the JOBS→MECHANICAL JOB PARAMETERS→ RESTART menu.

| Help | |
|---|---|
| Volume C | ADD RIGID, CHANGE RIGID |

## 2. Auto Step Enhancements

Adaptive load and time step control is a powerful method to obtain accurate results for complex nonlinear simulation. The AUTO STEP option has been enhanced to insure that a numerical solution can be obtained even if physical or numerical instabilities occur. This improves the robustness for both quasi-static and dynamic analyses.

### A. New Quasi-Static Damping Algorithm

A new quasi-static damping algorithm is now available (Figure 5) to makes the automatic time stepping scheme more robust for problems involving sudden changes in behavior of the system (contact, instability, etc.). With this algorithm, the expected quasi-static damping energy will be a small fraction of the total expected strain energy in the model. The major advantage of this method is that the user does not have to specify any damping factor, unless he wants to change the small fraction (default $2.0 \times 10^{-4}$) of the total expected strain energy. The method can be combined with other time step control methods, e.g. plastic strain energy rate, incremental displacements, etc.

An alternative method has been implemented where the actual damping is not applied to the system, but the values are only calculated to control the time step This second method is not currently supported by the GUI, but is available under AUTO STEP option in *MSC.Marc Volume C: Program Input*. This means that an undamped static analysis is performed and no criteria have to be chosen to control the time step.

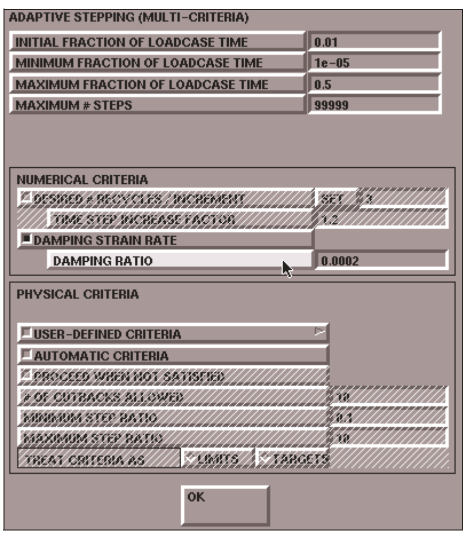


**Figure 5: Adaptive Stepping Menu**

### B. Scale Step Adjustment

When you use aggressive time step increase factor (values greater than 1.25) with the AUTO STEP algorithm, it is possible for the time step to increase rapidly due to initial linear behavior, but cut back sharply later due to sudden nonlinearities. In order to reduce this possibility, a more gradual scaling based on the ratio of actual number of recycles to the desired number is adopted when scale factors greater than 1.25 are specified by the user. When the actual number of recycles are greater than 60% of the desired number, the increase scale factor is limited to 1.25 for user-defined factors in the range of 1.25 to 1.5625. For the user-defined factors greater than 1.5625, the increase scale factor is limited to 0.8 times the given value.

## C. Damping to avoid Exit 3015

The AUTO STEP algorithm typically reduces the time step and repeats an increment when the increment fails to converge within a desired number of recycles. This process ends in exit 3015 if the minimum time step that you allowed is reached during the cutbacks, and convergence has not yet been achieved. To reduce the possibility of a premature exit due to the minimum time step being reached, a special damping scheme has been introduced.

This scheme is only used when you have not already flagged quasi-static damping to be used with the AUTO STEP algorithm. The scheme is triggered when the analysis is about to fail with exit 3015 for the first time. The increment is repeated with a time step of FCLRG times the minimum time step, where FCLRG is the user-defined maximum allowed time step increase factor (default is 10). Also, quasi-static damping is added into the system of equations for this trial and the damping factor is obtained by setting the predicted damping energy equal to a small fraction of the strain energy in the system. If this process fails the first time, the increment is repeated with a time step of $FCLRG^2$ (usually 100) times the minimum time step and with the quasi-static damping turned on. If this still fails, then the analysis ends with exit 3015. Also, it is not available in the very first increment of a loadcase since the damping factor is only calculated at the end of the first increment. Finally, once the damping to avoid exit 3015 is turned on, it remains on for the remainder of the loadcase.

| **Help** | |
|---|---|
| Volume A | AUTO STEP |
| Volume C | AUTO STEP |

# 3. Analysis Speed, Memory, and Scalability Enhancements

## A. Single Input File in Parallel Analysis

You can now run a DDM job using a single input file without the need for a preprocessor to generate the domains and to split the model data into input files for each domain. The decomposition into domains is performed within MSC.Marc. The single input file is read on one processor, the domain data is created and the filtered model files are passed to the other processors through MPI.

To run a DDM job using multiple input files after domain decomposition has been performed by MSC.Marc Mentat or MSC.Patran, include the command line argument `-nprocd` (or `-np`) followed by the number of domains.

There are three methods to run a DDM job using a single input file.

- Use the command line argument `-nprocds` (or `-nps`) followed by the number of domains. In this case, no changes are required to the input data.
- Include the PROCESSOR parameter in the input file with 1 in the 5th field. In this case, the command line argument `-nprocd` (or `-np`) could be used.
- Set the environment variable `nprocds` equal to the number of domains.

In the current release of MSC.Marc, the DDM single input file capability will not support any analyses types and input options not currently supported by the traditional multiple input files DDM.

The following model and history definition options and command line arguments are not supported in the MSC.Marc 2005 release.

- ACTIVATE and DEACTIVATE
- ACTUATOR
- BEGIN SEQUENCE and END SEQUENCE
- CHANNEL
- CONRAD GAP
- DAMPING
- DISP CHANGE if not FIXED DISP format. The DISP CHANGE option is supported only if the 1st entry of the data block following DISP CHANGE is equal to 0.
- EXCLUDE
- FXORD
- GLOBALLOCAL
- INSERT
- NEW
- POINT LOADS with follower forces
- SUPERPLASTIC
- TIME-TEMP
- TYING CHANGE
- UDUMP
- *def* command line option

- Auxiliary input files
- New table input format

The following additional capabilities are now available:

- Ability to generate a single post file from a DDM run
- Ability to use nonconsecutive element and node numbering

The following notes apply to the use of the single post file option:

- To generate a single post file from a DDM run, the 5th field of the 2nd data block under the POST option needs to be set to 2.
- Although general DDM jobs support post revisions 7 and higher, the ability to generate a single post file from a DDM job is restricted to post revisions 9 and higher.
- Reading a single post file back into a MSC.Marc DDM job using the `-pid` command line option is not generally supported in the 2005 release. It is only available for the AXITO3D and PRE STATE options.
- Single input file jobs with continuous post files, i.e. with the POST option after the RESTART option, both jobs should be run using either multiple post files or single post files. If multiple post files are used, then the number of domains should be the same in both jobs.

You can use the 6th field of the PROCESSOR parameter to choose the domain decomposition method. The following methods, also supported in the MSC.Mentat 2005 release, are available in the MSC.Marc 2005 release:

- Metis best domain decomposition (default)
- Metis element-based domain decomposition
- Metis node-based domain decomposition
- Vector domain decomposition
- Radial domain decomposition
- Angular domain decomposition

| **Help** | |
|---|---|
| User's Guide | Running DDM job |
| Volume A | Domain Decomposition |
| Volume C | PROCESSOR, POST |

## B. Improved Speed And Memory

There are a few areas the performance has been substantially improved for large problems over the years in MSC.Marc. The speed and memory improvements herein have been obtained by addressing the various important aspects of analysis: element (displacement based and Herrmann) and material behavior (plasticity and rubber), contact (rigid and deformable), and solvers (direct and iterative). One measure of performance is the speed and memory required to run the same problem from version to version. There are three problems that have been run in versions 2000, 2001, 2003, and 2005 that span nearly five to six years. They show continuing improvement in the performance of the simulation software.

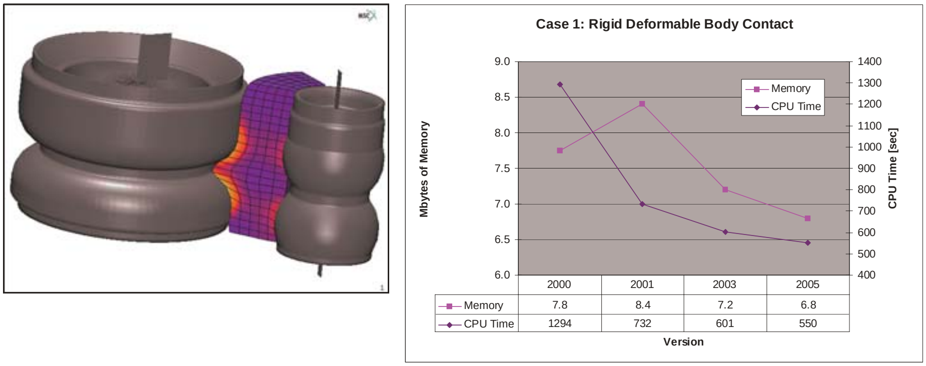


**Figure 6: Case 1: Rigid-Deformable Body Contact**

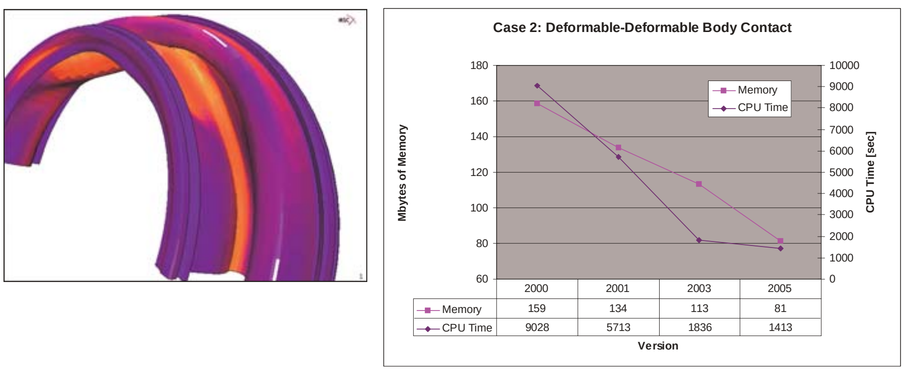


**Figure 7: Case 2: Deformable-Deformable Contact**

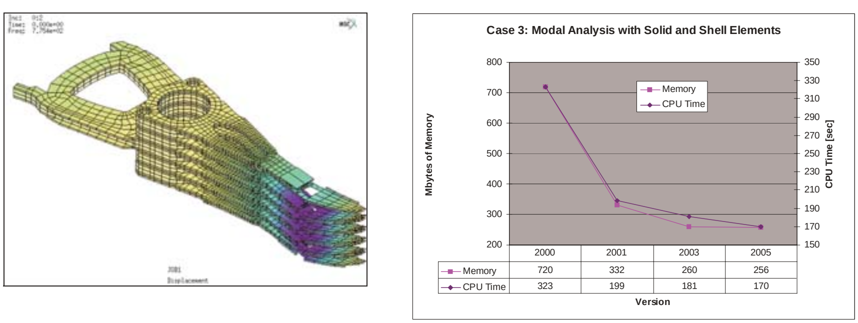


**Figure 8: Case 3: Model with Solid and Shell Elements**

| Help | |
|---|---|
| User's Guide | Analysis Performance Improvements |

# 4. Element Technology

## A. New Elements for Magnetostatics Analysis

MSC.Marc 2005 offers three new elements that you can use in a magnetostatic analysis. These elements are a 4-node and a 10-node tetrahedral element, and a 2-node line element. With these new tetrahedral elements, it is possible to use automatic meshers which will facilitate meshing complex structures. The purpose of the line element is to define an external loading, so this element does not have material or geometric properties.

Menus for selecting these new elements in MSC.Marc Mentat are shown in Figure 9.

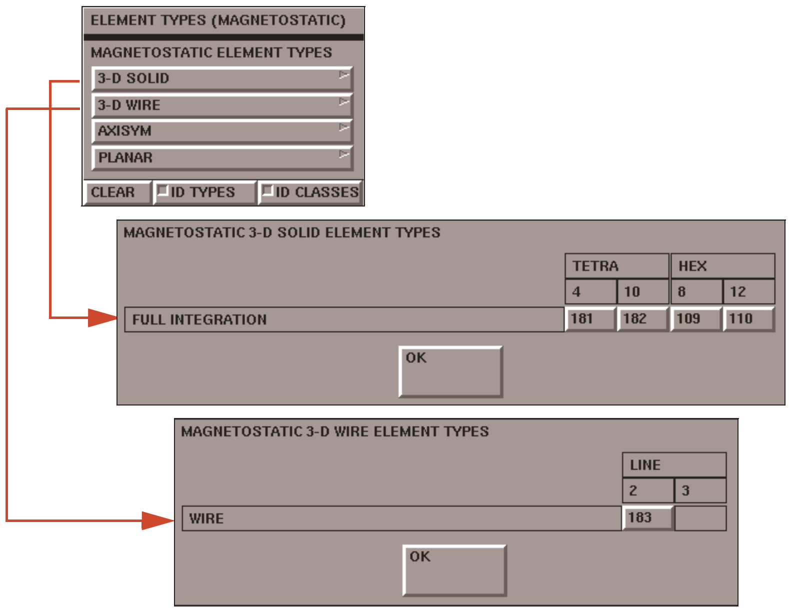


**Figure 9: Element Menu for the New Magnetostatic Elements**

You can place either place the line element on element edges of the solid elements or embedded in a host material. The direction of this current is along the direction of the line elements. Figure 10 shows the magnetic field around a coil in air modeled with 10-node tetrahedral elements, and line elements embedded in the host elements using the INSERT option. Looking closely at the figure, the line elements can be seen between the strong gradient of the magnetic induction.

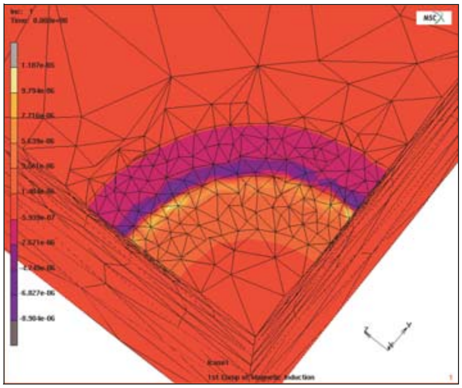

**Figure 10: Contour Plot of the X-direction of the Magnetic Induction around a Coil**

Figure 11 shows the menu for selecting boundary conditions in a magnetostatic analysis.

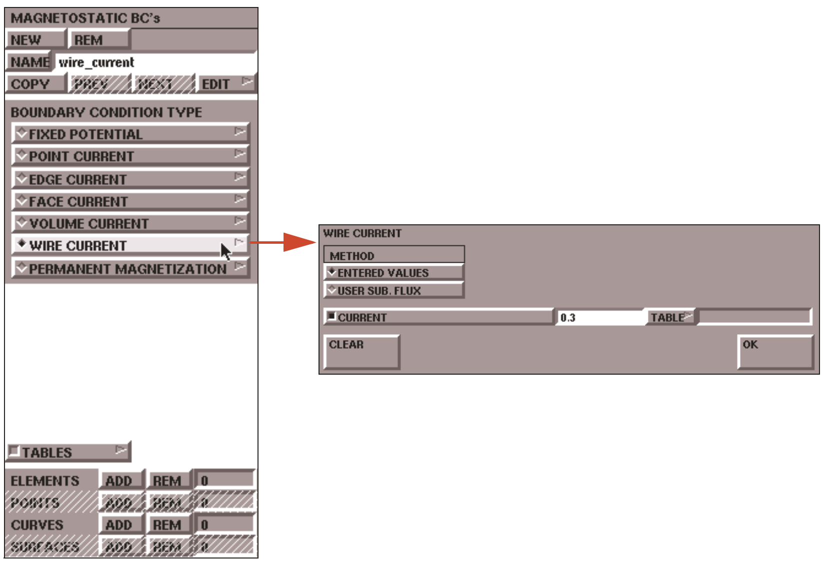


**Figure 11: Boundary Condition Menu for Wire Current**

| **Help** | |
|---|---|
| User's Guide | New Magnetostatic Elements |
| Volume B | Element 181<br>Element 182<br>Element 183 |
| Volume E | 8.87 |

## B. Rebar Enhancement

In many cases, the stiffness of reinforcing cords is much smaller in compression than the stiffness in tension. This is the so-called "micro-buckling" behavior.

Post code 487 has been added to enable the postprocessing the angle between rebar and the projection of reference axis on the rebar layer plane.

REBAR and POST model definition options have been modified to accommodate the above enhancements.

| **Help** | |
|---|---|
| Volume C | REBAR, POST |

# 5. Ease-of-Use and General Enhancements

## A. PRE STATE

The PRE STATE option allows you to:

- Specify initial conditions based on existing MSC.Marc results stored in ASCII (t19) or binary (t16) post files.
- Transfer data from a two-dimensional analysis to a three-dimensional analysis. This includes:
  - Transfer from axisymmetric analysis to 3-D (the old AXITO3D option is merged into the new PRE STATE option)
  - Transfer 2-D plane-strain analysis to 3-D
  - Transfer 2-D generalized plane-strain analysis to 3-D
- Transfer data on a per contact body basis from an old model to a new model.

The following capabilities are supported:

- Multiple contact body names
- Parallel processing
- Support models not created by MSC.Marc Mentat
- Nodal data types that are transferred include displacements, velocities, accelerations, and temperatures
- Element data types that are transferred include stresses, strains, thermal strains, creep strains, plastic strains, equivalent stresses, and strains.

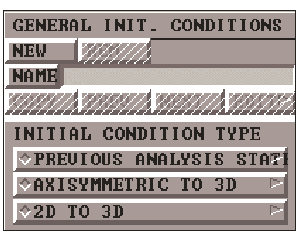


**Figure 12: Support of the PRE STATE Options under General Initial Conditions**

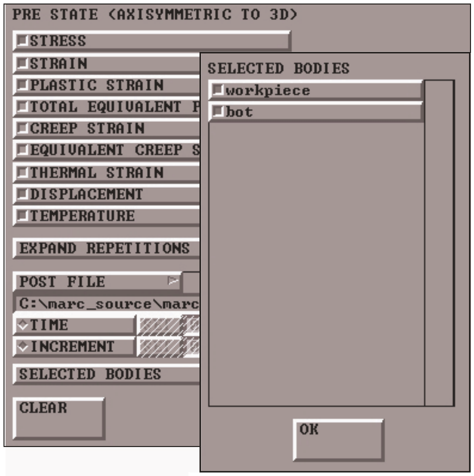


**Figure 13: Select Data Type and Contact Bodies to Transfer**

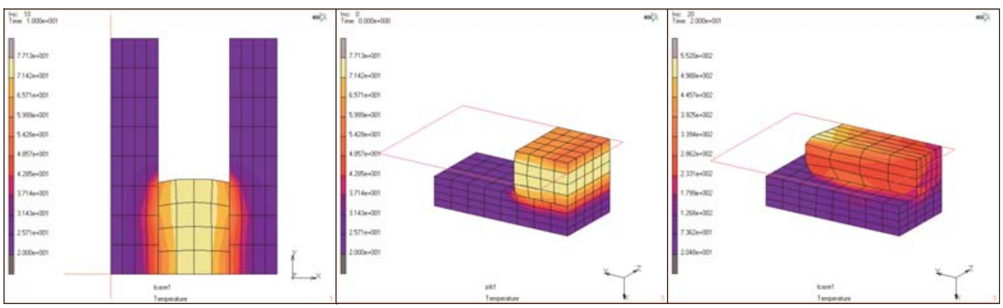

**Figure 14: Multiple Body Transfer from 2-D to 3-D**

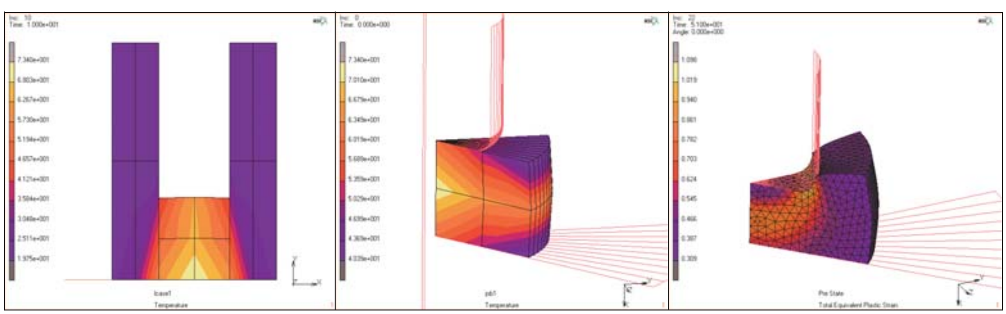

**Figure 15: Transfer from Axisymmetric to 3-D**

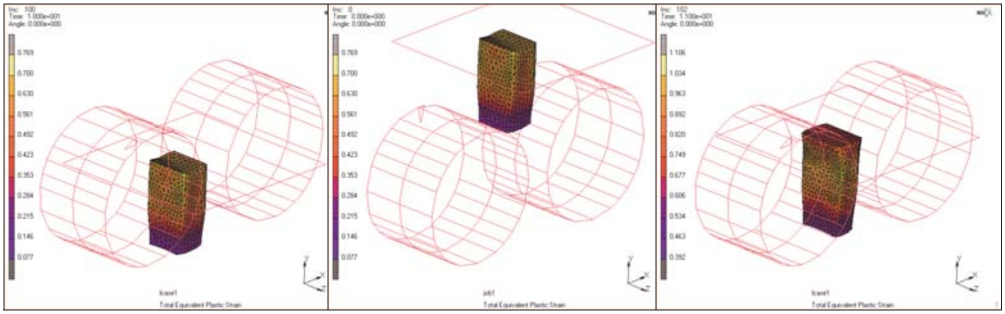

**Figure 16: Transfer from 3-D to 3-D**

| **Help** | |
| --- | --- |
| User's Guide | PRE STATE in Machining Simulation |
| Volume A | PRE STATE |
| Volume C | PRE STATE |
| Volume E | 8.86 |

## B. Structural Zooming Analysis

Local variations, such as changes to the model geometry or increasing mesh refinement to achieve a better evaluation of the local gradients in the solution, often require a complete re-analysis of the entire model. However, in cases when these local changes have negligible influence on the solution (a certain distance away from the changes), it is computationally more efficient to model only the part (with the local changes) and its vicinity. This can be achieved by applying the existing loads or/and boundary conditions in the local model along with a group of properly defined kinematic conditions to the local boundaries which connect the local model to the global model.

The new GLOBALLOCAL option in MSC.Marc 2005 enables you to perform this type of analysis. A typical MSC.Marc structural zooming analysis contains two steps.

(1) A global run to obtain global results.

(2) A local run to define kinematic boundary conditions in the local model and to obtain refined results in the local model.

This procedure can be repeated as many times as desired with any local analysis being the global analysis of next level refinement.

The GLOBALLOCAL option is used in the input of the local run to define the list of nodes connecting to the global model. MSC.Marc will automatically calculate the deformation (temperature) history of these nodes, based on their locations in the global model and on the solution of the global analysis. The obtained deformation (temperature) history is then applied to the nodes as prescribed kinematic boundary conditions to establish a complete local model.

The global to local modeling can be used in the following cases:

| Global Model | Local Model |
|---|---|
| 2D Solid | 2D Solid |
| 3D Solid | 3D Solid |
| 3D Shell/Membrane | 3D Shell/Membrane |
| 3D Shell/Membrane | 3D Solid |

This procedure may be used in linear or nonlinear analyses, and steady state or transient heat transfer analyses.

### MSC.Marc Input

A new model definition option GLOBALLOCAL is required to use this capability.

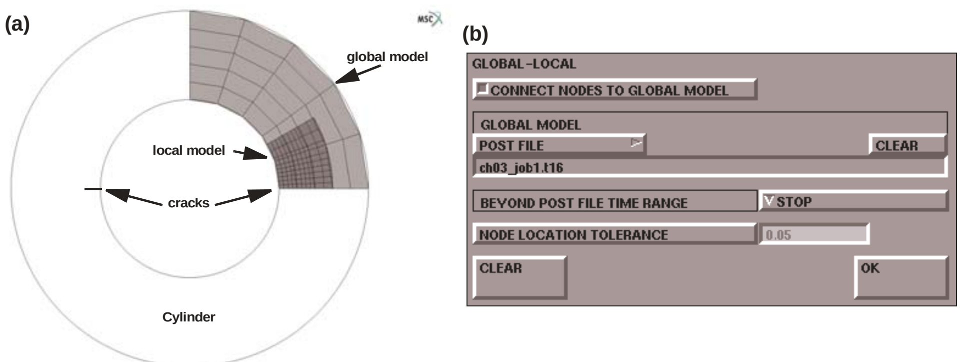


**Figure 17: (a) Structural Zooming Analysis (b) GLOBAL-LOCAL Menu**

| Help | |
|---|---|
| User's Guide | Analysis of a Manhole with Structural Zooming |
| Volume A | Globallocal |
| Volume C | GLOBALLOCAL |
| Volume E | 8.88 |

## C. Point Load Follower Forces

A follower force capability for point loads has been added to MSC.Marc 2005. The features are:

- Allow MSC.Nastran style support for follower force with point loads. The capability is similar to the FORCE1/FORCE2, MOMENT1/MOMENT2 bulk data options in MSC.Nastran. The magnitude of the load or moment is specified and the direction to be used for each nodal load is explicitly given through a list of 2 nodes or 4 nodes in the POINT LOAD option. This capability is supported through the SOL 600 module of MSC.Nastran.

- Allow automated support for follower force with point loads. When this option is used, the user specifies the load vector as usual. MSC.Marc then automatically determines an optimal nodal vector. The load vector direction is then constantly updated such that the initial angle between the load vector and the optimal nodal vector is maintained. This option allows you to specify the follower force option without extra effort. This option is directly supported by MSC.Marc Mentat.

The following features are available.

- Support is available for follower force in serial as well as in parallel using DDM or single input format. When the DDM scheme is used for the Nastran style follower force, care should be exercised that the nodes used to specify the direction are in the same domain in which the node having the point load applied.
- A global flag on the FOLLOW FOR parameter has to be set to indicate that point loads in the model can possibly be follower forces. Once this is set, you have additional freedom to indicate on the POINT LOAD option if an individual point load is a follower force or not.

The limitations of the feature are as follows.

- The follower force feature for point loads is not available for harmonic loads or Fourier loads.
- Point load vectors specified through user subroutine FORCDT cannot be used with the follower force option.
- Only the automated MSC.Marc style of follower force is supported by MSC.Marc Mentat. The MSC.Nastran style of follower force can be used via SOL 600 or by editing the input deck.

A planar cantilevered elasto-plastic beam is subjected to a point load that follows the geometry. The deformed configuration of the structure showing the updated load direction at the end of the loading phase is shown in Figure 18.

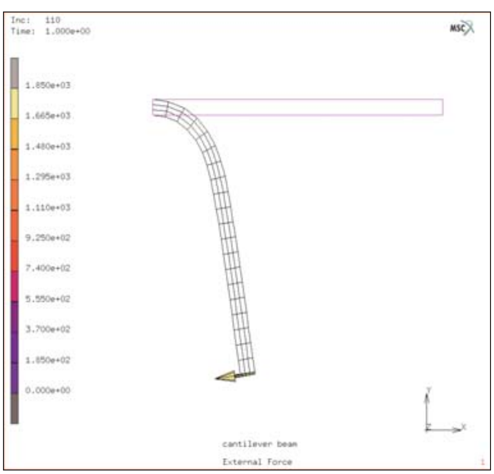


**Figure 18: Deformed Configuration at end of Loading Phase showing Updated**

### Direction of Point Load

| Help | |
|---|---|
| Volume A | Follow force |
| Volume C | FOLLOW FOR, POINT LOAD (with TABLE input), POINT LOAD (without Table input) |
| Volume E | 3.41 |

## D. Table Input

The MSC.Marc 2005 release offers the capability to define boundary conditions, material properties, and contact behavior through the use of table format. There are two aspects to this capability.

(1) Data can be provided via very general table input.

(2) Boundary conditions may be applied to geometric entities (points, curves, and surfaces) and remain associated with these entities.

This input format is advantageous because it permits:

(1) Easier definition of complex behavior.

(2) Reduces the need for user subroutines.

(3) Permits analyses that were not feasible with previous versions.

(4) Allows distributed loads to be applied on boundaries of bodies that are globally remeshed in 2-D.

(5) Improves the connectivity with CAD software and other MSC products.

(6) Reduces the number of iterations for some analysis.

(7) Reduces the size of the input file.

To activate the new input format, use NEW-STYLE TABLE button on the JOBS→RUN menu, as shown in Figure 19.

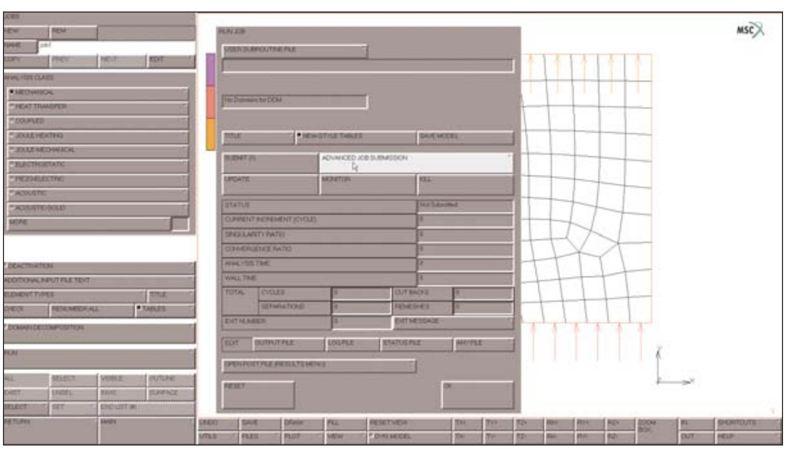

**Figure 19: JOBS→RUN Menus**

## Definition of Tables

The definition of tables is similar to previous versions in MSC.Marc Mentat, with the following critical differences.

1. A table may have four independent variables such as $(x, y, z, t)$, or $\bar{\varepsilon}^{pl}$, $\dot{\bar{\varepsilon}}^{pl}$, $t$ (see Figure 19).

2. The number of independent variable types has been increased. There are currently over 50 independent variable types. Figure 20 shows the menu used to initiate a multi-dimensional table, and Figure 21 shows a table with two independent variables.

3. The table may be defined by a mathematical equation (see Figure 22).

4. The table data or equation is directly transferred into MSC.Marc, and is evaluated at run-time by MSC.Marc.

The tables are defined in a menu similar to MSC.Marc Mentat 2003 version. When creating a new table, you will first be prompted for the number of independent variables. If there is only one independent variable, the behavior will be identical to the previous release. All old model files and data files can be used with the new release.

**Figure 20: Activate a New Table and indicate the Number of Independent Variables**

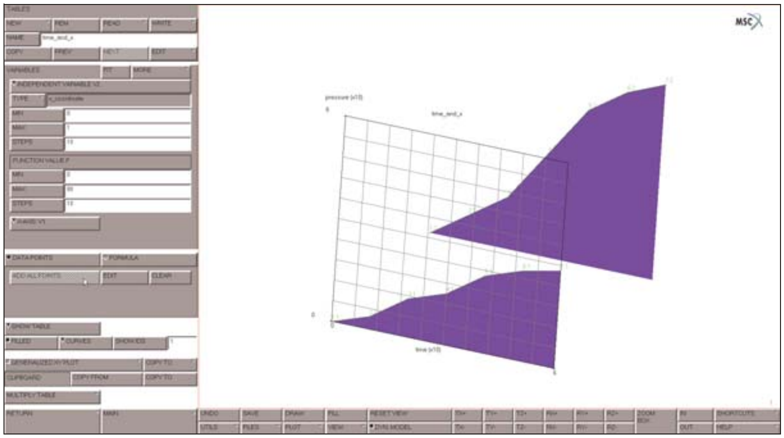

**Figure 21: Table to define Pressure as a Function of Time (1st independent variable) and X-coordinate Location (2nd independent variable)**

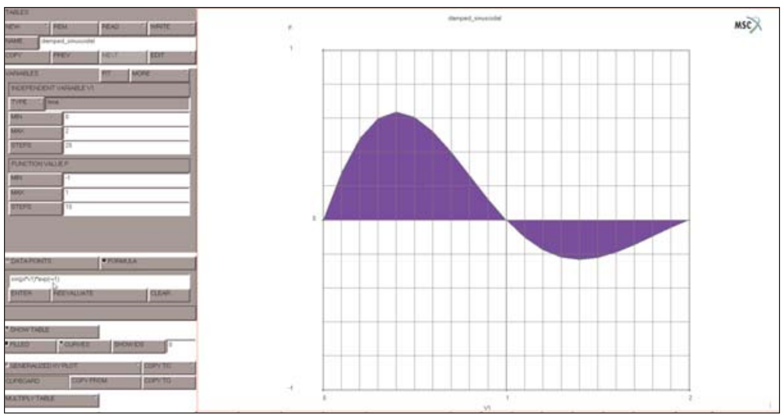

**Figure 22: Table defined with Mathematical Function for Damped Sinusoidal Behavior**

**Note:** Table evaluation will occur in MSC.Marc based upon the time at the end of an increment. Function shown is $\sin(\pi t) * e^{-t}$.

## Application of Tables

### Boundary Conditions

The tables may be associated with all boundary conditions in most analysis types. The additional capability is that these tables may now be a function of position, so a spatially varying load may be applied easily.

### Material Properties

Tables may be applied to virtually all material properties, under the assumption that the independent variable are physically meaningful. As an example, the Young's modulus can be a function of position and temperature to represent a nonhomogeneous, temperature dependent material. If the Young's modulus is made a function of the equivalent plastic strain, MSC.Marc will issue an error message, as this type of mechanics is neither valid nor supported.

As part of this development, more materials permit temperature dependent properties without the need to resort to user subroutines, such as

- Ogden coefficient
- Crack data

### Contact Data

Tables may be applied to most of the physical quantities associated with contact. This includes the velocity, coefficient of friction, and heat transfer coefficients. While the velocity could be defined before, the evaluation is now done within the program, so nonlinear prescribed motion is more accurately represented in conjunction with adaptive time stepping procedures (without implementing a user subroutine). Also, the coefficient of friction can now be a function of the normal stress, or in a coupled analysis, a function of the temperature.

### Welding

You may also use tables with the new welding capability in the MSC.Marc 2005 release to define the velocity of the heating device.

### Gaskets, Springs

These capabilities utilize the tables option identically to the previous release.

**Note:** In the MSC.Marc 2005 release, the table input cannot be used in Fourier, Fluid, and Bearing analyses.

### MSC.Marc Input

The input associated with this new capability is significantly different from the previous input and may be found in the *MSC.Marc Volume C: Program Input* manual. Existing Marc input files will continue to work with the new version. The new input style is initiated by the TABLE parameter option. All tables or mathematical equations are defined by the TABLE model definition option. All boundary conditions and initial conditions are modified to allow tables to be referenced.

Additionally, all boundary conditions and initial conditions have a name associated with them and may be defined before the END OPTION. A boundary condition is only active if it is referenced by the LOADCASE model definition (increment 0) or in the load history section. This allows better compatibility with the graphical user interfaces. All material property options ISOTROPIC, ORTHOTROPIC, MOONEY, OGDEN, etc. have also been modified, along with the CONTACT and CONTACT TABLE options to permit the use of tables. Because the motion of a contact surface may be easily defined, the MOTION CHANGE option is no longer required.

Table driven input may be generated using either MSC.Marc or MSC.Patran.

| **Help** | |
|---|---|
| User's Guide | New-style Table Input<br>Radiation Analysis<br>Application of BC on Geometry with Remeshing<br>Glass Forming of a Bottle with Global Remeshing |
| Volume A | Throughout manual |
| Volume C | TABLE (parameter), TABLE |
| Volume E | 3.42, 4.20, 7.34, 8.92, 8.98 |
| Volume E, Demo_Tables Directory | e2x35. e2x41, e3x2, e3x3, e3x4, e3x5, e3x7, e3x8, e3x9, e3x10, e3x11, e3x12, e3x14, e3x15, e3x17, e3x19, e3x20, e3x21, e3x26, e3x27, e3x28, e3x29, e3x31, e3x33, e3x34, e3x35, e3x36, e3x37, e3x40, e4x2, e4x3, e4x5, e4x6, e4x11, e4x12, e4x13, e4x17, e4x18, e4x20, e5x8, e5x9, e5x12, e5x14, e6x1, e6x14, e6x22, e7x2, e7x4, e7x11, e7x17, e7x18, e7x19, e7x20, e7x21, e7x22, e7x25, e7x27, e7x29, e8x4, e8x7, e8x10, e8x12, e8x15, e8x16, e8x25, e8x26,e8x31, e8x38, e8x39, e8x42, e8x45, e8x46, e8x47, e8x48, e8x50, e8x51, e8x53, e8x54, e8x55, e8x59, e8x60, e8x61, e8x62, e8x65, e8x67 |

## E. Geometrically Applied Boundary Conditions

MSC.Marc Mentat and MSC.Patran have supported the application of boundary conditions on geometric entities for many years. However, in the MSC.Marc 2005 release, the geometric information is passed into MSC.Marc, along with the attach (associativity) information between the geometry and the finite element model. This offers the possibility for boundary conditions to be automatically reapplied after global remeshing for 2-D analysis. In the future, this will be expanded to 3-D.

There are two aspects of this from a user perspective.

(1) Verification that the associativity is correct between the finite element mesh and the geometry.

(2) Application of the boundary conditions to the geometry.

There are no fundamental change in how the ATTACH option works between MSC.Marc Mentat 2003 and 2005 releases, but the following briefly reviews the capabilities.

The ATTACH option permits:

- Nodes to be attached to points.
- Element edges to be attached to curves.
- Element faces to be attached to surface.

The AUTO MESH capabilities permit the following.

- 2-D planar meshers: attach edges to curves, and nodes to the endpoints of curves
- Surface mesher: attaches faces to surface, edges to curves, and nodes to the endpoints of curves
- Solid mesher: if the bounding elements are attached to surfaces, the created mesh will also be attached to the surface.

Duplicate

- If the combined duplicate approach is used, the new mesh and geometry will be simultaneously duplicated, and the attach information will be preserved.

Change Class

- All attach information is preserved. Newly created midside nodes will be placed on the geometric entity.

Convert

- Convert points to nodes: the nodes will automatically be attached.
- Curves to elements: the element edges will automatically be attached to the curve.
- Surfaces to elements: the element faces will automatically be attached to the surface.

Expand

- If a combined expand is done of a curve and the line elements associated with it, then a surface and shell elements will be generated, and the shell elements will be attached to the surface.
- If an expand of a 2-D element which is attached to a surface is expanded, the association between the element face and surface is lost. A solid is not created by the expand command.

Move

- If a combined move of geometric and finite element entities is made, then the attach information is preserved.

Relax

- Does not influence attach properties.

Revolve

- If line elements are attached to a curve that is revolved, only a surface is created, not shell elements.

Stretch

- Preserves any attach information.

Subdivide

- If a line element is attached to a curve, then, when the element is subdivided, the new elements are attached to the curve.
- If the edges of 2-D or 3-D elements are attached to a curve, when the element is subdivided, the newly created edges are also attached.
- If the faces of a 3-D element are attached to a surface, when the element is subdivided, the newly created faces are also attached.

Symmetry

- Preserves all attach information.

### Applying Boundary Conditions

The user aspect within MSC.Marc Mentat remains the same, but the resultant Marc input file is substantially different. In using the old input format, the boundary conditions would be converted to finite elements and nodes. With the new format, one will observe that geometric information is written to the input file with this information referenced in the boundary condition options FIXED DISP, POINT LOADS, DIST LOADS, etc.

Additionally, all mechanical load boundary conditions (POINT LOADS or DIST LOADS) are based upon defining total values, not incremental values. This is a result in change of behavior when user subroutine FORCEM and FORCDT is used. For more details, see *MSC.Marc Volume D: User Subroutines and Special Routines*. The prescribed displacements are either total values, or values relative to the beginning of the loadcase.

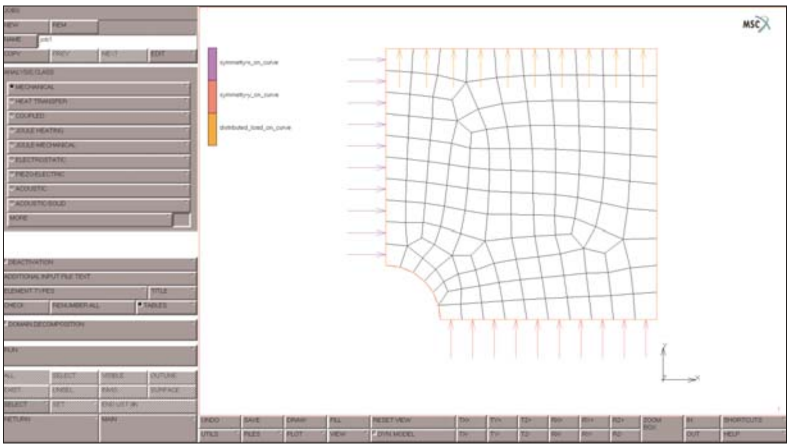

**Figure 23: Prescribed Displacements and Distributed Load applied to a Curve**

| **Help** | |
|---|---|
| User's Guide | New-style Table Input<br>Application of BC on Geometry with Remeshing |
| Volume A | Attach |
| Volume C | Throughout manual |
| Volume E | 3.42, 4.20 |

## F. Fracture Mechanics

The LORENZI J-Integral evaluation now supports large deformations and large strains (elastic as well as plastic). Contact between crack faces is also allowed. For the 2-D linear, elastic case the stress intensity factors for mode I and II are automatically calculated.

| **Help** | |
|---|---|
| Volume A | Mode separation |

# 6. Global Remeshing and Adaptive Meshing

## A. Loads and Boundary Conditions in Global Remeshing

Adaptive Global remeshing is a powerful capability that enables a new well formed mesh to be created after significant material distortion has occurred in an updated Lagrange analysis. It may be used with either elastic-plastic or large strain rubber material. In previous releases, the technique was limited to problems where the loads were either volumetric or applied via contact. The technique has been enhanced in two-dimensional analyses such that distributed loads and nodal boundary conditions are reapplied to the model after remeshing occurs. This may be used in conjunction with the advancing front quadrilateral or triangular mesher or the Delaunay triangular mesher at this time.

The boundary conditions may be directly applied to the mesh and work with:

- Point Loads on nodes
- Distributed Loads on element edges
- Fixed Displacements on nodes
- Point Flux on nodes
- Distributed Flux on element edges.

The boundary conditions may also be applied to the geometry entities and work with:

- Point Loads on points
- Distributed Loads on curves
- Fixed Displacements on points.

Both approaches require new table style input format.

The finite element mesh generator will insure that the new mesh contains a node at the same location as the nodes that has a point load or fixed displacement, and that elements span the same location in space if a distributed load is applied. The same is true if thermal boundary conditions are applied in a coupled analysis.

As an example, a glass forming is analyzed with an internal pressure to simulate the blow forming process (see Figures 24 through 26). A fixed displacement condition is applied to the top of the bottle in addition to the internal pressure. Thermal-mechanical coupling is assumed in the analysis. As shown in the figures, the internal pressure is maintained through the multiple remeshing steps required for this analysis.

#### MSC.Marc Input

As the new input style is used, no new input options are required.

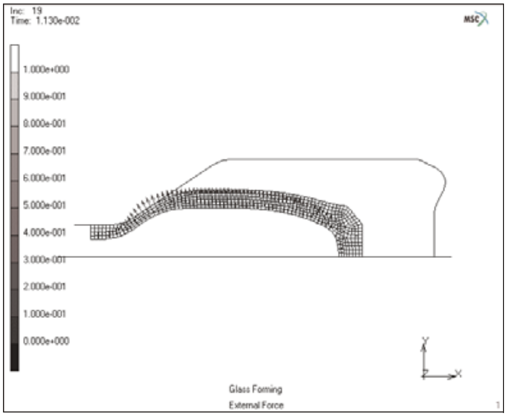


**Figure 24: Glass Forming Showing External Forces**

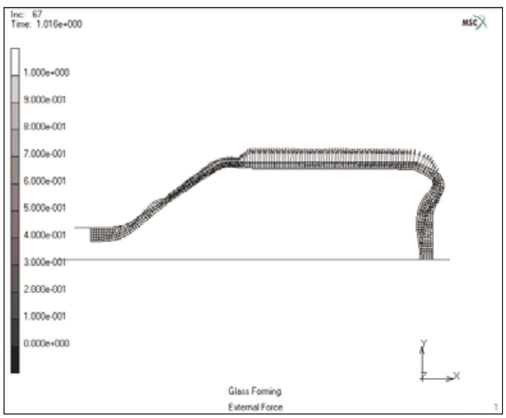


**Figure 25: Glass Forming Showing Final Shape of the Glass Container**

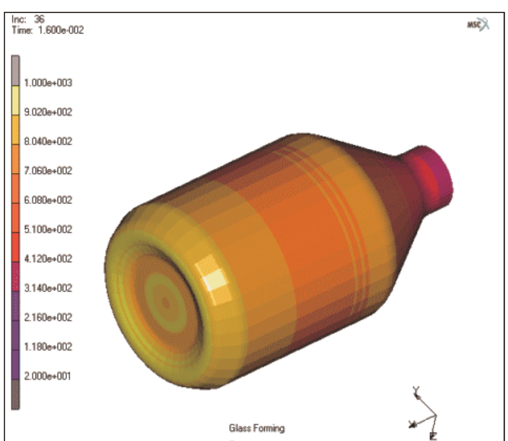


**Figure 26: Blow-Formed Glass Container with Temperature**

| Help | |
|---|---|
| User's Guide | Application of BC on Geometry with Remeshing<br>Glass Forming of a Bottle with Global Remeshing |
| Volume A | Remeshing with Boundary Conditions |
| Volume C | TABLE (parameter), ATTACH EDGE, ATTACH FACE, ATTACH NODE |
| Volume E | 8.91, 8.92, 8.96, 8.98 |

## B. Parallelized Local Adaptive Meshing

Enhancements have been made to support of local adaptivity in parallel where the previous restriction of no splitting of elements on the domain boundaries has been removed.

Figure 27 illustrates the use of this feature where a model with local adaptivity is analyzed using eight domains. A thin walled structure is subjected to bending and twist using rigid contact bodies. The original mesh density for each quarter is five by ten elements. A stress based adaptivity criterion is used.

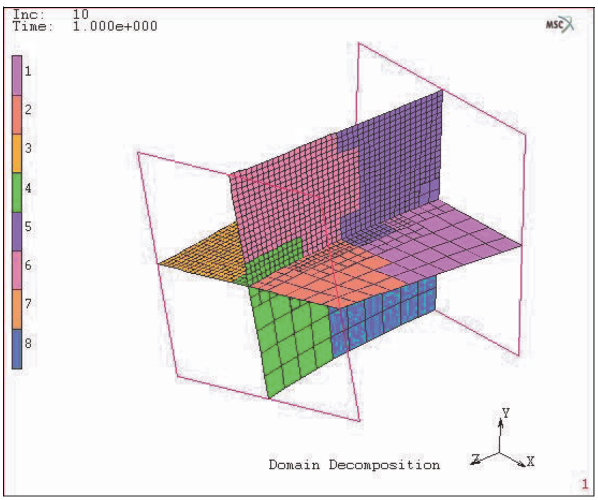


**Figure 27: Eight Domain Model with Local Adaptivity**

### Other Enhancements to Local Adaptivity:

(1) It is no longer necessary to specify the upper bound to the number of nodes and elements.

(2) Parts of the code have been rewritten to be faster. This is important for large models.

### MSC.Marc Input

No changes in the MSC.Marc input file are required to activate this capability.

| Help | |
|---|---|
| User's Guide | Parallelized Local Adaptive Meshing |

# 7. Manufacturing

## A. Machining

Basic capabilities for machining or material removal have been previously added in MSC.Marc version 2003. In the MSC.Marc release, a number of additional enhancements have been made to improve accuracy, computational efficiency, and user-friendliness.

The following improvements have been made in the MSC.Marc 2005:

1. The support of the CYCLE statement has been expanded from CYCLE/DRILL only to CYCLE/(DRILL, DEEP, TAP, BORE, and CBORE).
2. More efficient and accurate cutter-mesh intersection detection has been implemented.
3. Loadcase time synchronization is now allowed so that time-dependent contact and user boundary conditions can be used in conjunction with machining.
4. Cutter visualization is allowed during postprocessing of simulation results.
5. Local adaptive remeshing is added for NC machining analysis. For NC machining simulation purpose, this feature has been further enhanced so that multi-level element refinement, regular and irregular adaptive remeshing are possible.
    a. **Multi-level refinement of an element:** This allows an element to be subdivided into the maximum allowed sub-division levels within one increment.
    b. **Regular adaptive remeshing:** Elements that are partially intersected will be subdivided at each increment. Note that this can cause model size and computational time to increase significantly.
    c. **Irregular adaptive remeshing:** Elements that are partially intersected are not subdivided during the first coarse stage of machining. These elements are subdivided during a second fine stage of machining, wherein all the splitting is conducted in the last increment of the loadcase. This two-stage remeshing process can significantly reduce the computational time and memory usage.
6. **Automated residual stress import:** Based on the source of residual stresses, MSC.Marc can accept stress input data from MSC.Marc result files obtained from previous numerical analyses or residual stress data saved in text format data file from experimental analyses. Three methods have been made available in MSC.Marc to import residual stresses into the model prior to machining:
    a. **Pre State:** This method directly transfers data from a previously obtained MSC.Marc post file into the new MSC.Marc machining model.
    b. **Text data file:** By this method, MSC.Marc reads in the residual stress data stored in a text format data file. These stresses are automatically mapped into the FE model by MSC.Marc.
    c. **Table format:** Using this method, you can define the residual stresses as tables defined in space.

MSC.Marc Mentat supports all the improvements made on the solver side.

1. Load history definition interface (Figure 28a);
2. Adaptive remeshing definition interface (Figure 28b);
3. Residual stress input interface (Figure 29).

### MSC.Marc Input

There are changes in the DEACTIVATE History Definition, LOCAL ADAPTIVE, and INITIAL STRESS model definition options.

(a) (b)

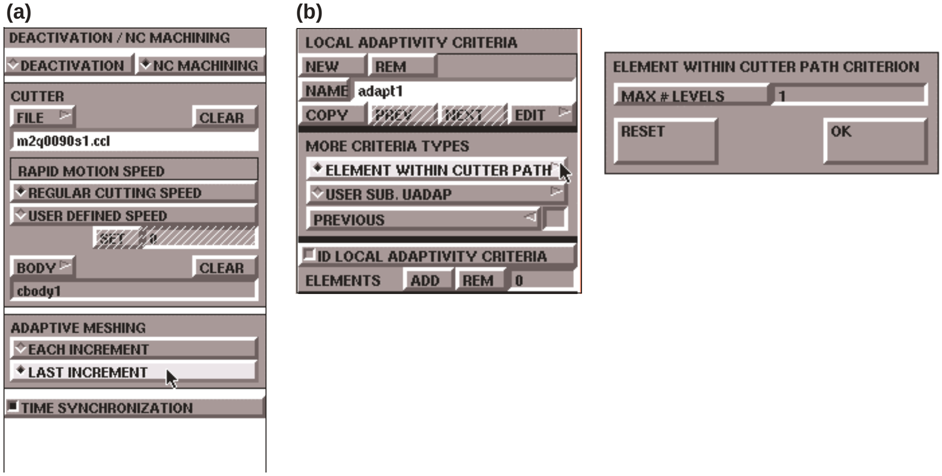


**Figure 28: (a) Deactivation/NC Machining Menu and (b) Adaptivity Criteria Submenu**

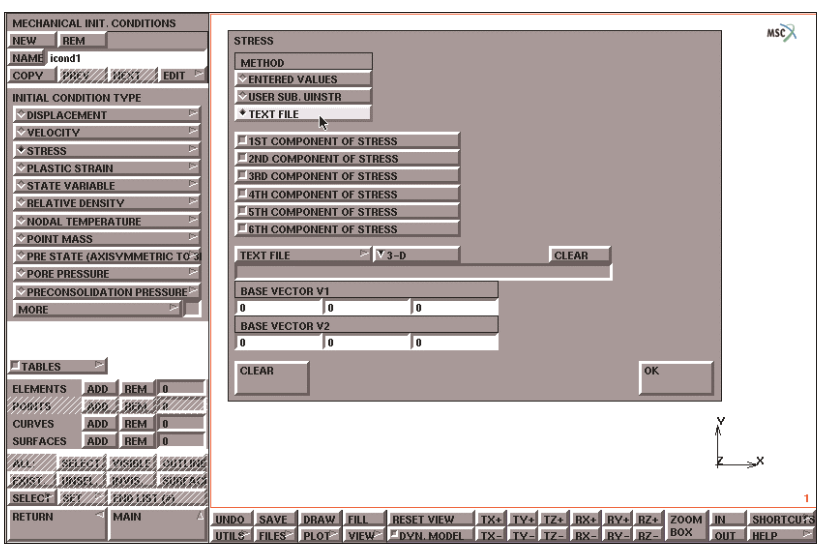


**Figure 29: Mechanical Initial Stress Menu**

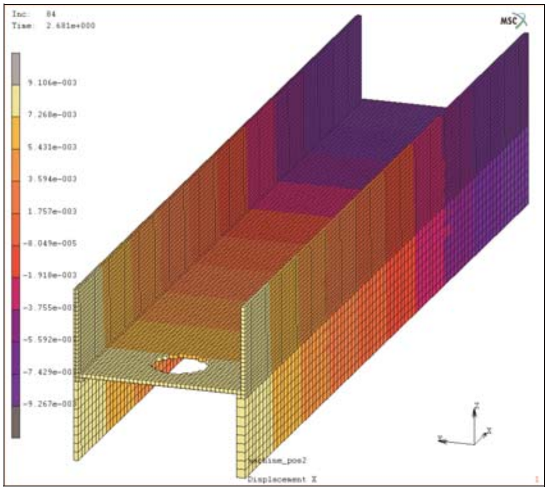


**Figure 30: Example of a Machined Part**

| **Help** | |
|---|---|
| User's Guide | FEM Simulation of NC Machining and PRE STATE |
| Volume A | NC Machining Processes |
| Volume C | MACHINING, DEACTIVATE, ADAPTIVE, INIT STRESS |
| Volume E | 8.89 |

## B. Welding

A number of enhancements have been made to facilitate welding simulation in MSC.Marc 2005:

- Standard heat source models like the disc source and the double ellipsoidal source are directly supported through the WELD FLUX option. User subroutine UWELDFLUX can be used for arbitrary weld heat fluxes.
- A number of options are available for specifying the weld heat source motion and for the weld arc orientation through the WELD PATH option. User subroutine UWELDPATH can be used for arbitrary weld paths.
- The automated creation of weld fillers and associated thermal boundary conditions are supported through the WELD FILL option. Both deactivated and quiet filler element techniques are supported. Special features are provided to facilitate convenient setup of multi-pass welds.

The following features are supported:

- Support is available for welding analysis in serial as well as in parallel using DDM.
- Both fixed stepping (TRANSIENT NON AUTO) and adaptive stepping (AUTO STEP) schemes are supported for welding simulations. Time step cutbacks are used in conjunction with the AUTO STEP scheme to ensure that default/user-defined temperature controls are satisfied.

- The welding capability is available for 2-D planar, axisymmetric, 3-D continuum, and shell elements. Both thermal and thermo-mechanical coupled analyses can be conducted.
- Local adaptive meshing using an automated box definition based on the current weld pool is available. An unrefine capability can also be used to coarsen the mesh once the heating source has moved away.
- The total weld heat input calculated by the program and the filler element activation history can be optionally printed out in the output file using the PRINT,31 parameter.

The MSC.Marc Mentat menus for the (a) weld flux, (b) weld path, and (c) weld filler are shown below:

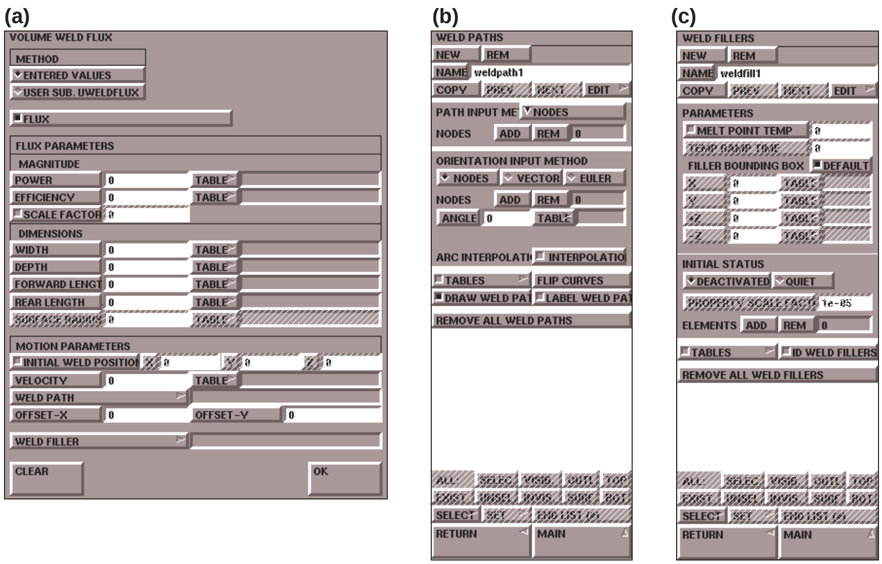


**Figure 31: Welding Menus**

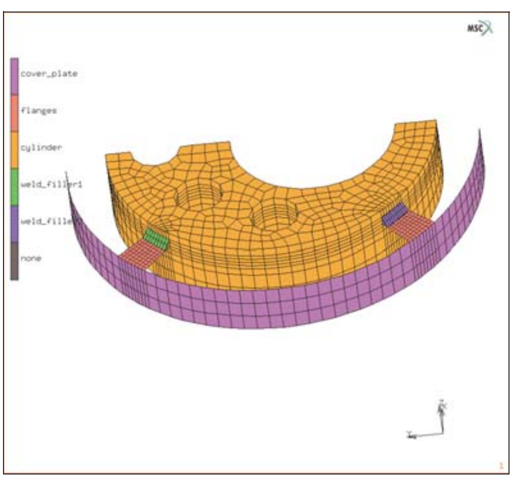


**Figure 32: Finite Element Mesh of Cylinder-Plate Joint**

### MSC.Marc Input

The WELDING parameter is used to initiate the simulation. WELDFLUX, WELDPATH, and WELDFILL model and history definition options control the heating, motion, and fill material respectively.

| Help | |
|---|---|
| User's Guide | Arc Welding Process Simulation |
| Volume A | Welding |
| Volume C | WELDING (parameter), ADAPTIVE, WELD FILL, WELD FLUX, WELD PATH |
| Volume D | UWELDPATH, and UWELDFLUX |
| Volume E | 8.93 |

# 8. Materials

## A. Shape Memory Materials

The following enhancements have been made to the shape memory alloy material capability.

**1.** Support of one- and two-dimensional elements: Both beams and shell elements can now be modeled besides the continuum (Figure 33) elements.

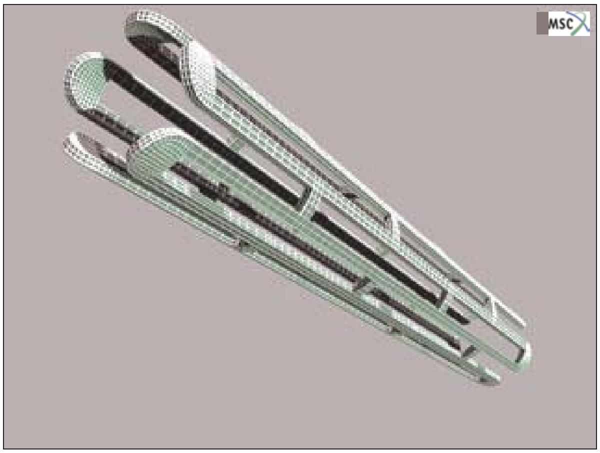

**Figure 33: Stent Model**

**2.** TRIP strains are available on the post file.

**3.** For the mechanical shape memory model (Aurrichio's model), additional data regarding the reference temperature, martensite and autenite slopes, and initial martensite volume fraction can now be input. This allows the analysis to be done at various temperatures (see Figure 34).

**4.** Improved accuracy of the stress integration by means of a new iteration method. The implementation of this algorithm prevents the overshoot observed in certain cases at the turning points in the shape memory curve.

**5.** Equivalence of the input (Conversion table given in *MSC.Marc Volume A: Theory and User Information Manual*) between the mechanical and thermo-mechanical models allows the use of either of the models for mechanical analysis. Also, the model parameters are better explained and parametric sensitivity of various input parameters is shown through various examples.

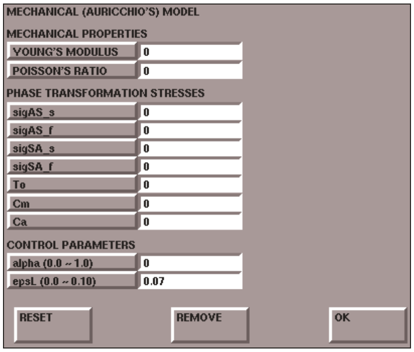


**Figure 34: Shape Memory Material Menu for Mechanical (Auricchio's) Model**

| **Help** | |
|---|---|
| Volume A | Shape Memory |
| Volume C | SHAPE MEMORY |
| Volume E | 8.81 |

## B. Generalized Strain Energy Function Model

A framework, based on the updated Lagrangian formulation, has been set up for hyperelastic material models. Within the framework, users can easily define their own generalized strain energy function models through a user subroutine UELASTOMER. This user subroutine will work for both the invariant-based and the principal-stretch-based strain energy functions, and work for energy functions with or without volumetric-deviatoric split.

This feature will allow users in rubber and foam industries to implement their own models in a very easy manner, and will enable MSC.Marc's global remeshing/rezoning capability for the foam applications where large deformations are usually involved.

The user subroutine UELASTOMER replaces existing user subroutines such as UENERG, UMOONY, UOGDEN, and UPSTRECH. These subroutines are supported in the product for backwards compatibility with older input files but they will not be supported in future releases.

For incompressible rubber-like materials, a mixed formulation which treats hydrostatic pressure as an independent variable, has been used in MSC.Marc. A linear relationship between pressure and volumetric strain is assumed in the mixed formulation. Therefore, only the deviatoric part of energy function needs to be defined in the user subroutine for rubber-like materials. Two types of rubber energy functions are supported:

(1) Invariant-based model, deviatoric part only
(2) Principal-stretch-based model, deviatoric part only.

For compressible foam materials, four types of generalized strain energy functions are supported within the framework:

(3) Invariant-based model
(4) Principal-stretch-based model
(5) Invariant-based model with volumetric and deviatoric split
(6) Principal-stretch-based model with volumetric and deviatoric split

MSC.Marc uses conventional displacement elements for compressible foam models.

### MSC.Marc Input

No new options are needed.

| Help | |
|---|---|
| Volume A | UELASTOMER |
| Volume C | MOONEY, OGDEN, FOAM |
| Volume D | UELASTOMER |
| Volume E | 7.23 |

## C. Large Strain Viscoelastic Foam Model

The large strain foam model has been expanded to support viscoelastic behavior, which is often required in the packaging industry.

The updated Lagrange technique is used to model viscoelasticity and damage in foam materials. For viscoelastic foam material, the VISCELFOAM option should be used.

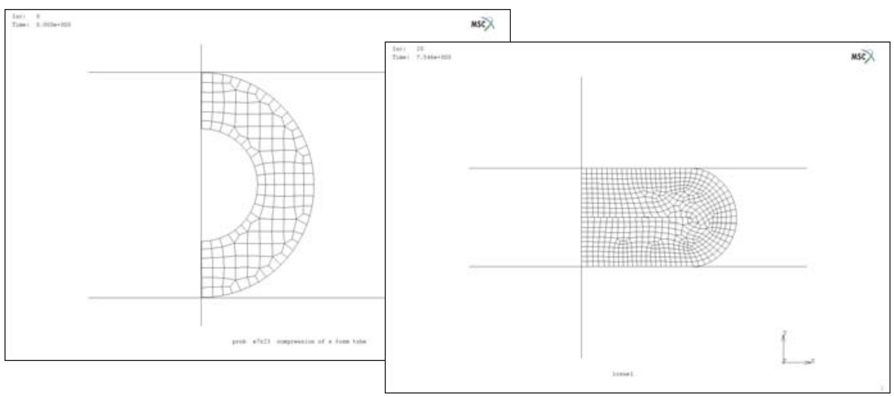

**Figure 35: Initial and End Mesh of Foam Analysis**

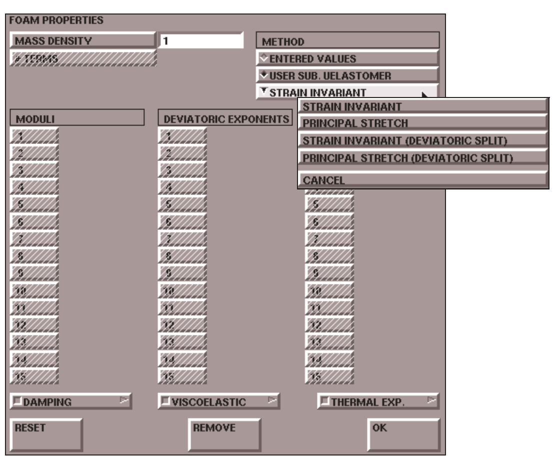


**Figure 36: Foam Properties Menu**

| **Help** | |
|---|---|
| Volume A | Viscoelastomer |
| Volume C | VISCELFOAM |
| Volume E | 7.23 |

## D. Hypoelastic Analysis

User subroutine HYPELA is no longer officially supported. You are encouraged to use the HYPELA2 user subroutine which has an expanded call list and is more powerful.

# 9. Multiphysics and Thermal Enhancements

## A. Radiation Enhancements

The accurate, fast calculation of viewfactors for radiation analysis is a critical component of thermal analysis. In many problems, especially where large temperature differences occur over space, radiation is the dominant mechanism for thermal transport. Radiation calculations are important for all space systems (satellites, airplanes, rockets, and space shuttle).

It is also important in manufacturing processes such as casting or where furnaces are involved. It has been gaining more importance in automotive applications because of the increased use of electronic devices under the hood.

The enhancements to the radiation capabilities in the MSC.Marc 2005 release include:

- A new viewfactor program which is both more accurate and faster than the previous Monte Carlo technique.
- Viewfactors are calculated from within the MSC.Marc program and may be recalculated if large deformation occurs. This also allows the modeling of articulated structures.
- The radiation calculation may be used in conjunction with local adaptive meshing. The viewfactors will be calculated if required.
- The radiation calculation may be used with global adaptive meshing in two-dimensional problems.
- The emissivity may be entered as a table of position, temperature, etc.
- The cut-off for switching between implicit radiation calculations and explicit calculation is based upon relative values, as opposed to absolute values. This is easier to control.
- Frequency dependent emissivity may be entered through the table option.
- The user can visualize the magnitude of the viewfactors.
- The emissivity can be applied as a surface option as opposed to a material property.

Examples of MSC.Marc Mentat menus used to invoke these new capabilities are shown in (Figure 37 through Figure 42).

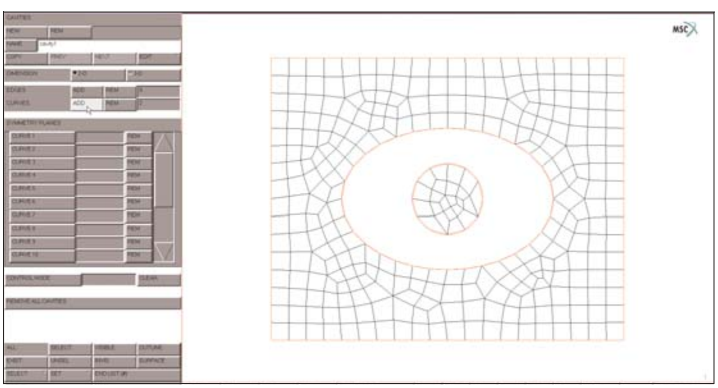

**Figure 37: CAVITIES Menu**

Define RADIATION CAVITY through MODELING TOOLS→CAVITIES Menu.

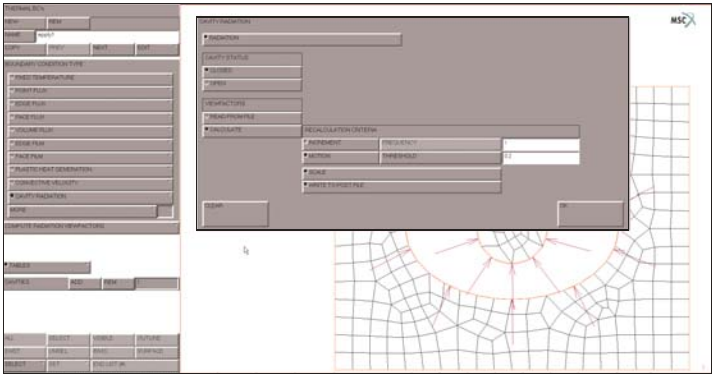

**Figure 38: CAVITIES RADIATION Menu**

Add CAVITY RADIATION BOUNDARY CONDITION through BOUNDARY CONDITIONS→ THERMAL→CAVITY RADIATION menu. Activate CALCULATE to perform viewfactor calculation in this analysis. Activate either OPEN or CLOSED CAVITY. Write viewfactors to post file for visualization.

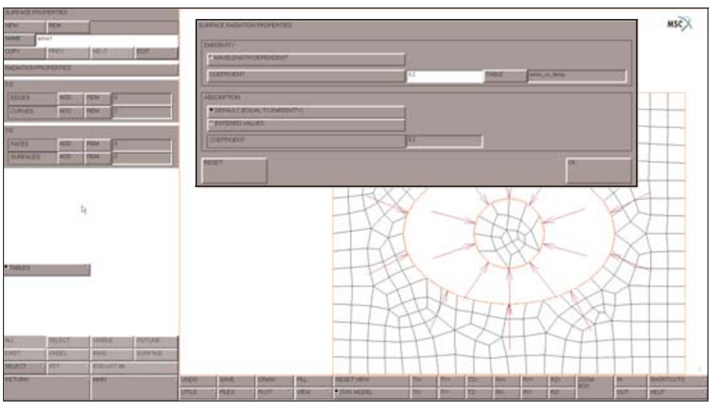

**Figure 39: SURFACE RADIATION PROPERTIES Menu**

Define the surface emissivity via MATERIAL PROPERTIES→SURFACE PROPERTIES→ RADIATION PROPERTIES menu. Note the temperature dependence on emissivity via table.

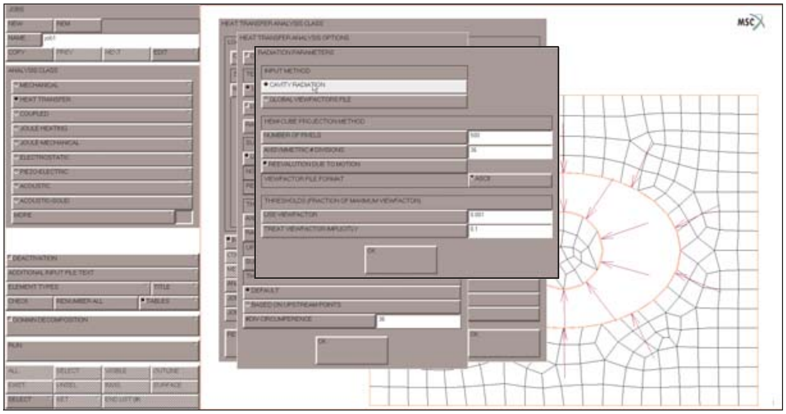

**Figure 40: RADIATION PARAMETERS Menu**

Use the JOBS→HEAT TRANSFER→ANALYSIS options to set the parameters for calculation of the viewfactors and cut-off values for implicit-explicit calculation. It may also be necessary to use the JOBS→HEAT TRANSFER→JOB PARAMETERS→UNITS and CONSTANTS menu to define the thermal units used in the analysis.

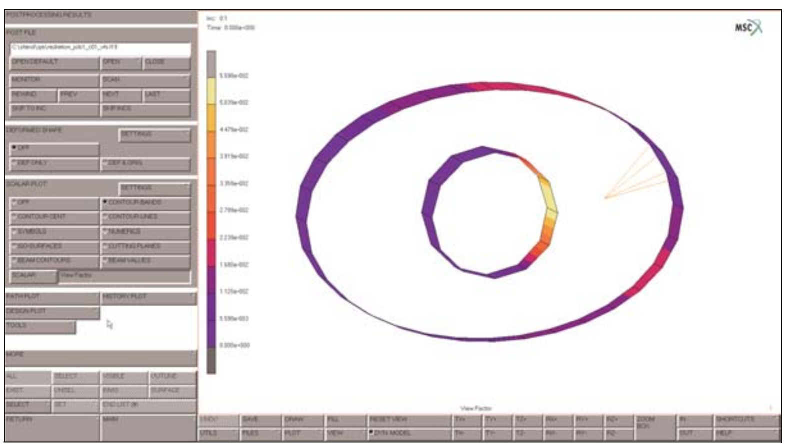

**Figure 41: POSTPROCESSING RESULTS Menu**

Open the *jid.vfs* post file containing the viewfactors. The pyramid indicates what would be the emitting face, and the colors indicate the respective viewfactors. Notice that the center cylinder results in a shadow.

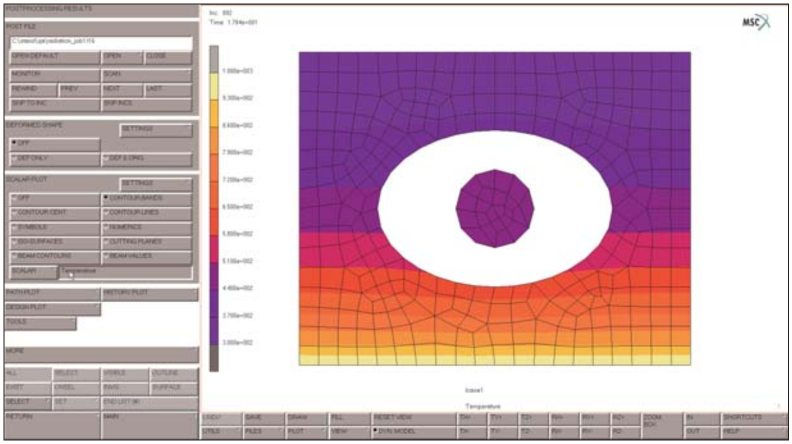

**Figure 42: POSTPROCESSING RESULTS Menu in Final Stage**

With this menu, you see the thermal results, including heating of internal cylinder.

### MSC.Marc Input

The use of the new radiation capability requires the RADIATION parameter. This is also used to define the accuracy of the calculation and the units used. The cavity geometry is defined in the CAVITY DEFINITION option. Symmetry curves (2-D or surfaces (3-D) are defined using the CURVE and SURFACE options. The control of the radiation calculation (open or closed), frequency of calculation, and post file output is defined in the RAD-CAVITY option. The surface emissivity is defined via the EMISSIVITY model definition option. The new capability requires that the table driven input format is used.

| **Help** | |
|---|---|
| User's Guide | Radiation Analysis |
| Volume A | Modified Hemi-Cube |
| Volume C | RADIATION (parameter), CAVITY DEFINITION, RAD CAVITY, EMISSIVITY |
| Volume E | 5.15, 5.20 |

## B. Coupled Electrostatic-Structural Enhancements

MSC.Marc version 2005 offers coupled electrostatic-structural analysis especially useful for modeling MEMS (Micro Electro-Mechanical Systems) devices. In this type of analysis, forces between charged bodies are used in the structural pass, and the electrostatic field is computed using the deformed geometry. In general for an electrostatic analysis, we can distinguish between conductors and insulators. For example, with a capacitor, the plates are the conductor and the dielectric or air between the plates is the insulator. When a potential difference is applied between the plates, charge will form on the surfaces of these plates. The charge on the different bodies generate the forces between these bodies, the Coulomb forces. To obtain the charge on bodies, the concept of contact bodies is used and in the case of the capacitor example, the plates and the dielectric are separate contact bodies. The charge is then computed at the interface of two contact bodies. Note that it is necessary that the insulating contact body is touching the conducting contact body (the contact bodies on which a Coulomb force is acting are touching). The usual mechanical and electrostatic material properties need to be selected for the elements used. Also, it is possible to select electrostatic element types for elements that are not active in the structural pass (Figure 44). This can be useful for modeling air.

However, the body representing the air will need to be remeshed if a deformation occurs. Figure 45 shows the MSC.Marc Mentat menus for the loadcase options. This is similar to menus of other coupled analysis. Figure 46 shows the JOB option menus, which also resembles menus for other coupled analysis.

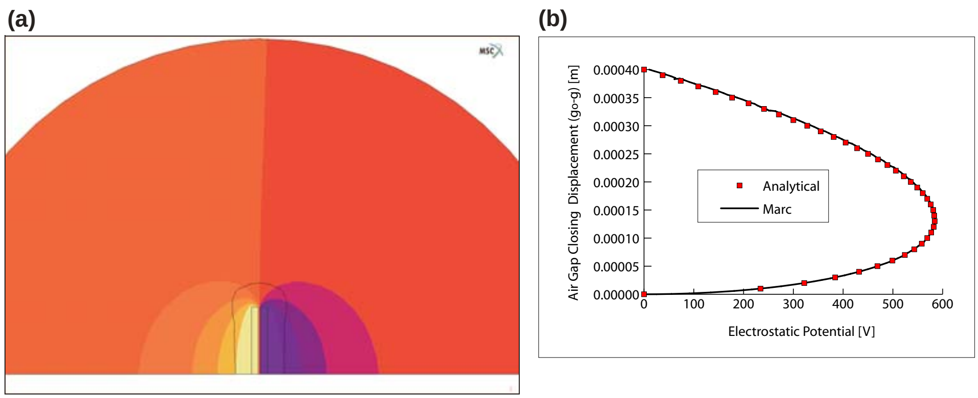


**Figure 43: (a) Electrical Potential Around Capacitor Plate (b) Potential as a Function of Gap Opening**

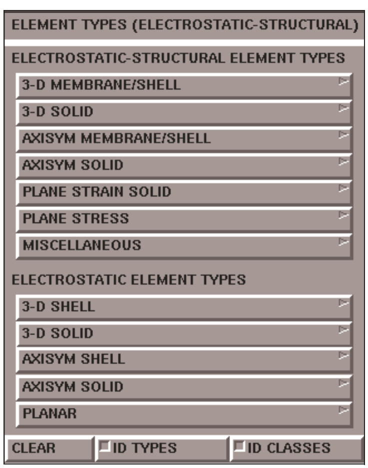


**Figure 44: Menu for Element Types**

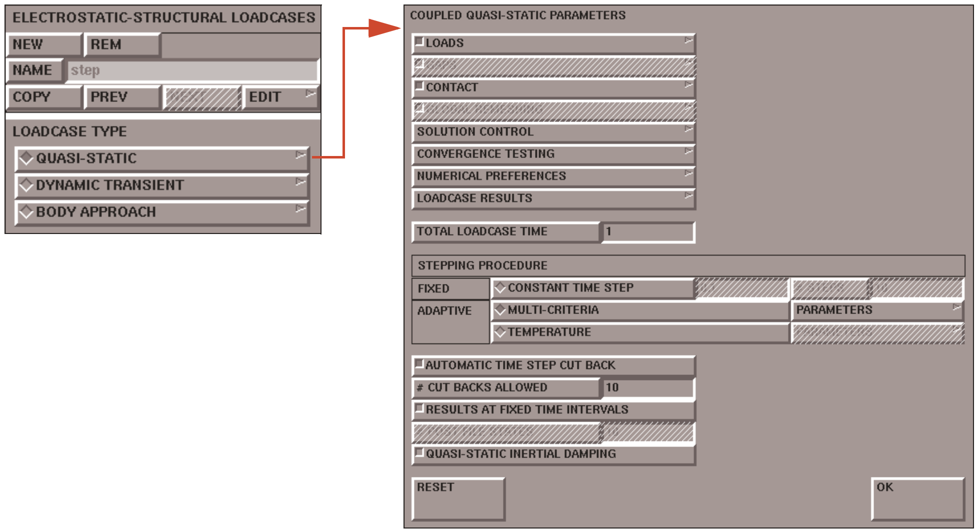


**Figure 45: Loadcase and Quasi-Static Parameters Menus**

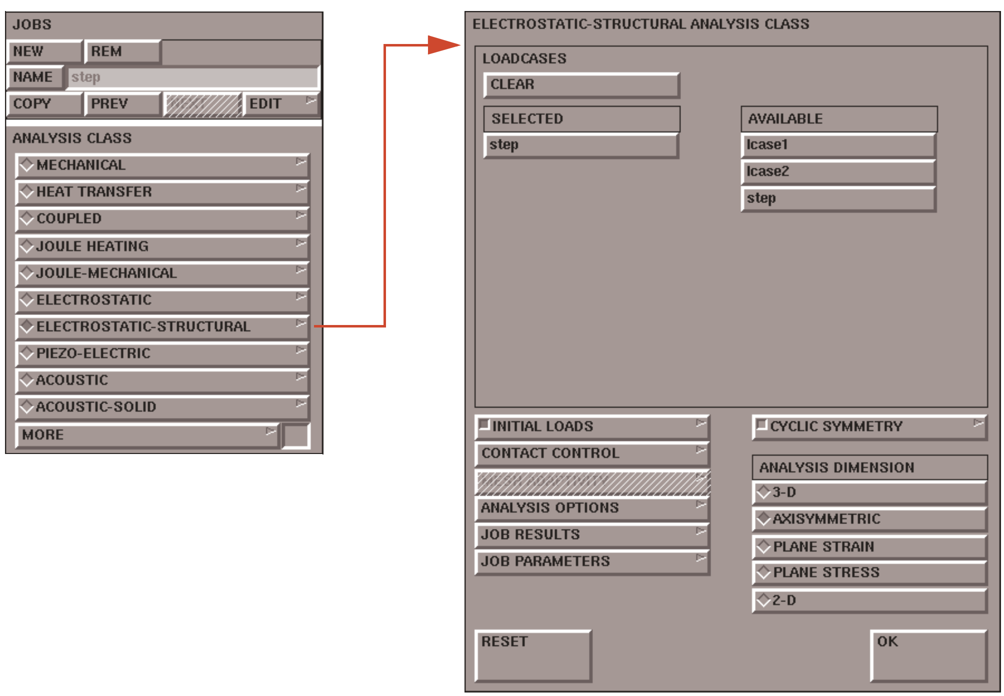


**Figure 46: Job Option Menu with Analysis Class Options**

### MSC.Marc Input

The STRUCTURAL and ELECTRO parameters are used to initiate this coupled analysis. The ISOTROPIC and other material options have been enhanced so both structural and electrical material data is input.

| Help | |
|---|---|
| User's Guide | Coupled Electrostatic Structural Analysis of a Capacitor |
| Volume A | Electrostatic-structural, electrostatic-structural loads |
| Volume C | STRUCTURAL (parameter) |
| Volume E | 8.94, 8.95 |

## C. Diffusion Analysis

The MSC.Marc 2005 release allows diffusion simulations to be performed. This is similar to the pore pressure capability, but is now available for all heat transfer elements. The user can define the permeability and porosity of the material and the viscosity, density and bulk modulus of the fluid. The pressure of the fluid is obtained.

| Help | |
|---|---|
| Volume A | Diffusion |
| Volume C | ISOTROPIC, ORTHOTROPIC, FIXED PRESSURE, POINT MASS, DIST MASS |

# 10. Multipoint Constraint Enhancements

## A. General Section Constraint

Enhancements have been made to the modeling of bolts and rivets in MSC.Marc and MSC.Marc Mentat. A new tying type (overclosure tying; code 69) has been implemented that can be used to create overlaps or gaps between two parts of a model. If the motion of these parts is constrained in the direction in which the overlap or gap is being created, then an overlap will introduce a tensile pre-stress in each of the parts and a gap will result in a compressive stress.

The new tying has one tied node and two retained nodes. The tied and first retained nodes are usually nodes on the boundaries of the two parts. The second retained node is most often a free node which is shared by all tyings which connect the two parts. It is also called the control node of the tyings, since it can be used to control the size of the overlap or gap:

(1) The displacement of the control node in a particular direction is equal to the size of the overlap or gap between the two parts in that direction; and

(2) The force on the control node is equal to the sum of the forces on the tied nodes of the tyings which share that control node and it is equal to minus the sum of the forces on the first retained nodes of these tyings.

The new tying type can be used in combination with contact. Any of the nodes of the tying can have a local coordinate system, defined by the TRANSFORMATION option. The tying reduces to tying type 100 (all degrees of freedom) between the tied and the first retained node in non-mechanical parts of the analysis.

Hence, a pre-stress can be defined in a bolt or a rivet by splitting the finite element mesh in two parts and connecting each pair of corresponding nodes on opposite sides of the split by overclosure tyings, such that the nodes of one part act as tied nodes, the corresponding nodes on the opposite side act as first retained nodes and all tyings share a common control node (Figure 47). Then, the total force on the side of the split where the tied nodes are located can be prescribed by applying a point load to the control node and the size of the overlap or gap can be prescribed by prescribing the displacement (change) of the control node.

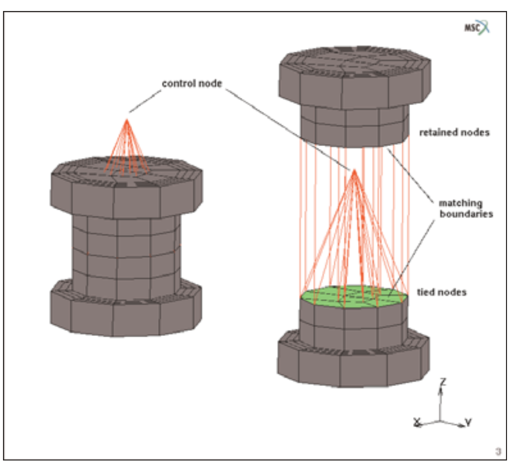


**Figure 47: Prestressing of a Bolt using Overclosure Tyings**

New menus are available in MSC.Marc Mentat to automatically split the finite element mesh into disjoint parts and for connecting the corresponding nodes on either side of the split with tyings of the new type (or any other type). The general procedure is as follows:

**1.** Split the finite element mesh of a bolt or rivet in two parts using the tools in the MODELING TOOLS→MATCHING BOUNDARIES menu (Figure 48a). The newly created boundaries on opposite sides of the split will be identified as matching boundaries and stored in the MSC.Marc Mentat model file.

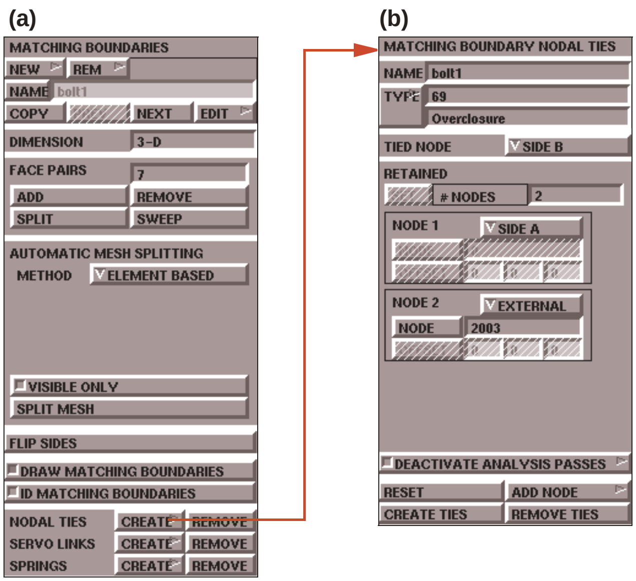


**Figure 48: (a) Matching Boundaries Menu (b) Matching Boundary Ties SubMenu**

**2.** Generate overclosure tyings between corresponding nodes of matching boundaries using the MATCHING BOUNDARY NODAL TIES submenu (Figure 48b).

### MSC.Marc Input

The TYING model definition option has been enhanced for this feature.

| **Help** | |
|---|---|
| User's Guide | Thermo-Mechanical Analysis of Cylinder Head Joint with Quadratic Contact |
| Volume A | Overclosure Tying |
| Volume C | TYING |
| Volume E | 8.89 |

## B. Large Rotation RBE3

The large rotation version of RBE3 in the previous version of MSC.Marc is based on an assumption of relatively small rotation increment. This limitation is eliminated in the current version. As shown in Figure 49, a stress free motion of a structure (solid elements are connected with shell elements using RBE3) is simulated nearly exactly using the current version.

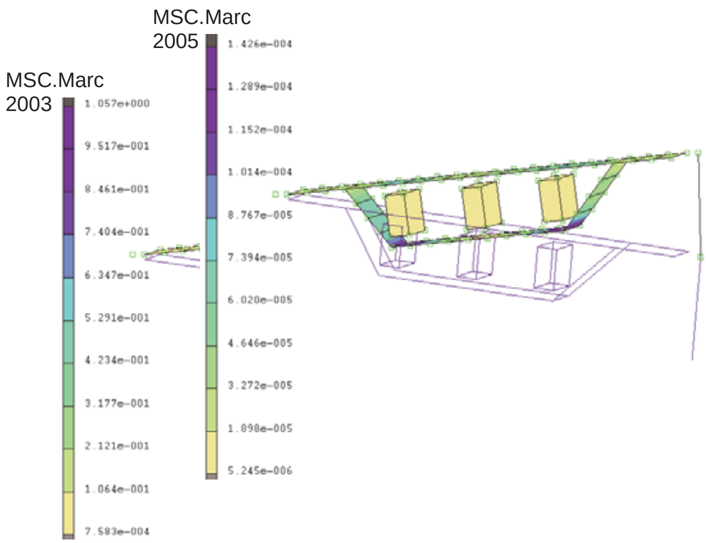


**Figure 49: Equivalent Stress Contour**

To enhance the compatibility with MSC.Nastran, two options are added in the current version. The first one is the ability for users to specify "UM" set as tied nodes. In this case, degrees of freedom of the reference node are not eliminated in the system matrices. By default, the reference node is used as tied node. The RBE3 menu for MSC.Marc Mentat is also updated to support this option as shown in Figure 50. For more detailed format of the new RBE3 option, please refer to *MSC.Marc Volume C: Program Input*.

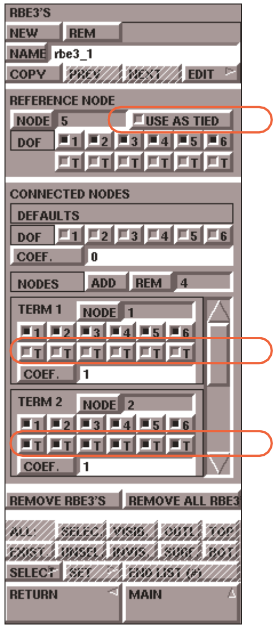


**Figure 50: New Menu for RBE3 with New Options**

The second option is the implementation of a "normalized" constraint for rotation degrees of freedom. The non-normalized formulation, implemented in MSC.Marc 2003, has a drawback that a RBE3 has unit dependence. The new normalized, formulation has no unit dependence and is set to default. To maintain downward compatibility, the non-normalized one can be activated using parameter RBE,,,2.

| **Help** | |
|---|---|
| User's Guide | RBE3 (General Rigid Body Link) |
| Volume A | RBE3 |
| Volume C | RBE3 |
| Volume E | 2.80, 4..19 |

## C. INSERT Enhancement

You can now define host bodies (contact bodies) in the INSERT option instead of host elements. A further enhancement is that if the host bodies are used for this option, it is allowed to remesh these host bodies. For example, this makes it possible to connect a spring to a body which is to be remeshed. It's also possible to insert a point load on a body to be remeshed.

Furthermore, you can now define shell or membrane elements as host elements. Be sure to properly define the shell thickness if you want to insert nodes/elements into shell or membrane elements.

The INSERT model definition option has been enhanced to accommodate the use of contact bodies.

# 11. Product Integration

## A. MSC.Adams – MSC.Marc Enhancements

### MSC.Marc – MSC.ADAMS MNF Interface

MSC.ADAMS/Flex allows flexible components to be included in MSC.ADAMS models to achieve more realistic simulation results. MSC.Marc 2005 is capable of generating a Modal Neutral File (MNF) representing the flexible component to be integrated into the MSC.ADAMS model. The flexible component can undergo linear or nonlinear loading in MSC.Marc up to the point where the MNF is requested to be generated. Generating a MNF from MSC.Marc is based on performing the most general method of Component Mode Synthesis (CMS) techniques, namely the Craig-Bampton method.

MSC.Marc allows the user to directly specify the interface nodes in the input. These are the nodes at which the flexible component may be connected to the rest of the MSC.ADAMS model or at which loads can be applied during the MSC.ADAMS simulation. MSC.Marc also allows automatic specification of interface nodes that come into contact with user selected contact bodies. This is very useful for some nonlinear analyses, such as tire footprint analysis in which the interface nodes are not known a priori.

The MNF generated by MSC.Marc contains the essential information for defining the component in MSC.ADAMS, such as the component's topology, list of interface nodes, generalized mass, generalized stiffness, and the Craig-Bampton mode shapes. It may also contain optional quantities, such as modal loads, modal preloads, and stress and strain modes.

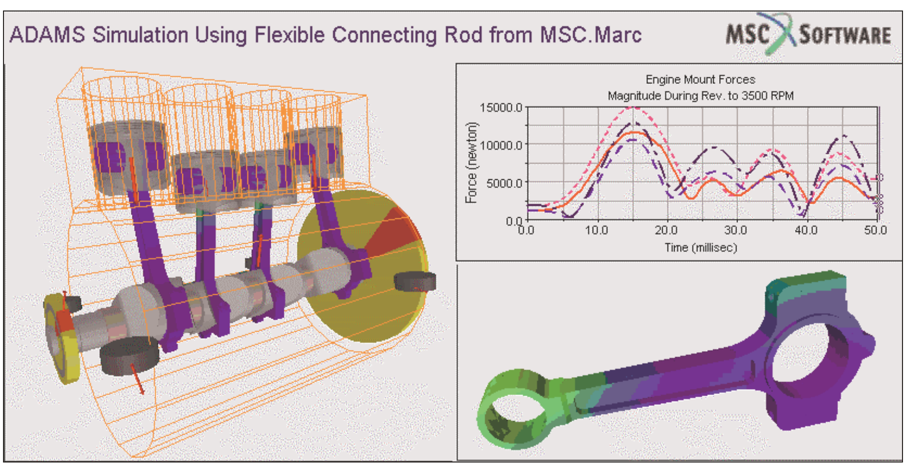


**Figure 51: Example of MSC.Marc and MSC.Adams MNF Interfacing**

#### MSC.Marc Input

The MNF parameter, MNF UNITS model definition, SUPERELEM model definition, and ASSEM LOADS history definition are used to activate these capabilities.

| **Help** | |
|---|---|
| User's Guide | MSC.Marc – MSC.ADAMS MNF Interface |
| Volume C | MNF, SUPERELEM, MNF UNITS, ASSEM LOAD |
| Volume E | 8.90 |

## 12. Changes in Defaults in MSC.Marc

1. **Contact bias factor**: The default has changed from 0 to 0.95. This yields improved behavior in friction as well as problems involving rigid body contact. In MSC.Marc Mentat this is accessible through the JOBS→MECHANICAL→CONTACT CONTROL→ ADVANCED CONTACT CONTROL menu.

2. **Auto Switch**: is now ON by default. This helps in switching convergence criterion where one type of criterion does not make sense due to physics of the problem to another one which is more meaningful. In MSC.Marc Mentat this is accessible through the LOADCASES→MECHANICAL→STATIC→CONVERGENCE TESTING menu.

3. **Automatic Loadstepping Parameter**: In AUTO STEP, the desired number of recycles have been increased from three to five. This is usually helpful in moderate to highly nonlinear problems in where five or more iterations (not counting the ones due to contact) can be needed and the time step may be frequently reduced due to a lower number of desired cycles per increment. In MSC.Marc Mentat this is accessible through the LOADCASES→ MECHANICAL→STATIC→MULTI-CRITERIA (PARAMETERS) menu.

4. **Inconsistent MPC treatment**: As opposed to the previous releases where the job was allowed to continue due to the way MPCs (TYING, RBE, SERVO LINKS, INSERT, CONTACT etc.) were listed in the input, MPC reordering is now done if the input file includes a VERSION,11 parameter by default. If reordering is not successful, an EXIT 2011 occurs. This yields a more accurate treatment of MPCs with better analysis results. This is controlled using the MCP-CHECK parameter. In MSC.Marc Mentat this is accessible through the JOBS→MECHANICAL→ANALYSIS OPTIONS→ADVANCED OPTIONS menu.

5. The following modifications have been made to the memory accounting:
   - With the SUMMARY option, a detailed accounting of memory usage is displayed at the end of each increment (or at a desired frequency), as well as at the end of analysis.
   - Without the SUMMARY option, total memory usage is displayed after each increment and at the end of analysis.

   The following modification has been made to the timing information.
   - A detailed accounting of the timing information is provided only when including the SUMMARY option.

6. **Radiation Viewfactor tolerances:** The tolerances associated with whether a viewfactor contribution will be treated implicitly or explicitly or neglected have changed from absolute tolerances to relative tolerances. Input files that include a Version 10 parameter will behave as before.

7. **Material identifiers:** The storage of material data has improved with this version, so that nonconsecutive material ids require no additional memory. But, this requires the user to explicitly specify a material id. Material id = 0 is no longer acceptable and an EXIT 13 will occur.

# MSC.Marc Mentat 2005

## 13. Modeling Shell and Membrane Structures

The modeling of shell and membrane structures in MSC.Marc Mentat has been enhanced. In the following, `shell` may also be read as `membrane`. The objective of these changes is to make it easier to apply boundary conditions on the top and/or bottom of shells, such as required for distributed fluxes, convective heating, and radiation in heat transfer analysis.

### Definitions

A 3-D shell geometric property (defining the shell thickness) can now also be assigned to surfaces, instead of assigning it directly to elements. All appropriate elements (quadrilateral and triangular elements) attached to these surfaces automatically inherit the geometric property.

An axisymmetric shell geometric property (defining the shell thickness) can now also be assigned to curves, instead of assigning it directly to elements. All appropriate elements (line elements) attached to these curves automatically inherit the geometric property.

Surfaces are considered to be “shell surfaces” if they have a 3-D shell geometric property. Shell surfaces have a top and a bottom side.

Curves are considered to be “shell curves” if they have an axisymmetric shell geometric property. Shell curves have a top and a bottom side.

Quadrilateral and triangular elements are considered to be “3-D shell elements” in the following cases:

- if they have a 3-D shell geometric property
- if they are attached to a shell surface
- if they have a 3-D shell MSC.Marc element type.

3-D shell elements have a top and a bottom side, also referred to as top face and bottom face.

Line elements are considered to be “axisymmetric shell elements” in the following cases:

- if they have an axisymmetric shell geometric property
- if they are attached to a shell curve
- if they have an axisymmetric shell MSC.Marc element type.

Axisymmetric shell elements have a top and a bottom side, also referred to as top edge and bottom edge.

### Drawing

A graphical representation of the thickness of shell curves and surfaces by drawing shell curves and surfaces in expanded mode is not yet available.

A graphical representation of the thickness of shell elements is now possible by drawing shell elements in expanded mode (both in pre- and postprocessing). This also allows for graphical picking of top and bottom sides of shell elements.

New commands added:

`*draw_shells_expanded`
Toggles the drawing of shell elements in expanded mode;
default = on.

See GEOMETRIC PROPERTIES→SHELL PLOTTING OPTIONS→PLOT EXPANDED or PLOT→ELEMENTS SETTINGS→SHELL PLOTTING OPTIONS→PLOT EXPANDED.

`*shells_expanded_pre_only`
If shell plotting in expanded mode is on, this command decides if this should be done in preprocessing only, or also in postprocessing;
default = on (preprocessing only).

See GEOMETRIC PROPERTIES→SHELL PLOTTING OPTIONS→PREPROCESSING ONLY or PLOT→ELEMENTS SETTINGS→SHELL PLOTTING OPTIONS→PREPROCESSING ONLY.

The thickness used for drawing a shell element in expanded mode is derived as follows:

- if the element has a shell geometric property, the thickness of that geometric property is used, else
- if the element is attached to a shell curve or surface that has a shell geometric property, the thickness of that geometric property is used, else
- a default thickness is used.

Both constant and variable thickness defined in a geometric property are supported.

Note that the shell normal direction in a node is computed as the average of the normals of the elements that the node belongs to. Currently, in case of T-joints, or when elements have opposite normals, no averaging is done.

New command added:

`*shells_expanded_thick`
Sets the thickness for drawing shell elements in expanded mode, only used if the thickness of the element has not been defined via a geometric property;
default = 0.1.

See GEOMETRIC PROPERTIES→SHELL PLOTTING OPTIONS→DEFAULT THICKNESS or PLOT→ELEMENTS SETTINGS→SHELL PLOTTING OPTIONS→DEFAULT THICKNESS.

## Commands

The introduction of the shell sides concept as described above has an influence on certain commands. These commands fall into three categories:

**1.** Commands that now accept top and bottom sides of shells as input and actually distinguish top and bottom:

- add/remove entities to/from cavities
- add/remove entities to/from surface properties
- add/remove entities to/from (`edge`) or (`face`) boundary condition types
- select edges, faces, curves, or surfaces (SINGLE or ASSOCIATION method)

- store edges, faces, curves, or surfaces in a set
- select entities using the PLANE method (requests a single element face).

2. Commands that now accept top and bottom sides of shells as input but treat top and bottom identically:
   - select entities by edges, faces, curves or surfaces
3. Commands that now potentially generate top and bottom sides of shells as output:
   - select edges, faces, curves, surfaces

   Note that in attach commands, no distinction is made between top and bottom sides of shells.

New wildcards (entity list shortcuts) have been added for commands that accept top and bottom sides of shells as input (**1**, **2**):

`ALL: TOP (all_top)`
generates a list of all entities of the requested type except the bottom sides of shells

`ALL: BOTTOM (all_bottom)`
generates a list of all entities of the requested type except the top sides of shells.

Note that all other wildcards will potentially include both top and bottom sides.

New selection filter commands have been added for the selection of edges, faces, curves, and surfaces (**3**):

`TOP (*select_filter_top)`
This command specifies that top filtering will be used by the select commands. With top filtering, only those entities that are not bottom sides of shells will be selected by the select commands.

`BOTTOM (*select_filter_bottom)`
This command specifies that bottom filtering will be used by the select commands. With bottom filtering, only those entities that are not top sides of shells will be selected by the select commands.

For backward compatibility of old procedure files, a command has been added:

`*set_shells_old`
Hidden command, that can be used when running old procedure files within the new setup for the modeling of shell structures to force them to run in the same way as they did in previous MSC.Marc Mentat versions.
default = off.

If this command has been used,
- bottom sides of shells will be ignored by commands that accept top and bottom sides of shells as input (**1**, **2**)
- bottom sides of shells will be not be generated as output by commands that potentially generate top and bottom sides of shells as output (**3**)

The following commands now accept top and bottom sides of shell surfaces as input:

1. `*add_cavity_surfaces`, `*remove_cavity_surfaces`,
   `*add_srfprop_surfaces`, `*remove_srfprop_surfaces`,
   `*add_apply_surfaces`, `*remove_apply_surfaces`,
   `*select_surfaces` (SINGLE or ASSOCIATION method),
   `*store_surfaces`, `*remove_set_entries`

2. `*select_nodes_surfaces`,
   `*select_edges_surfaces`,
   `*select_faces_surfaces`,
   `*select_points_surfaces`,
   `*select_trim_points_surfaces`,
   `*select_trim_curves_surfaces`

   A mixed list of non-shell surfaces and sides of shell surfaces may be given in these commands.

   For shell surface `s1`, the top side is given by `s1:0` (or `s1`) and the bottom side by `s1:1`. A non-shell surface `s2` is given by `s2` (or `s2:0`).

   Top or bottom sides of shell surfaces cannot yet be selected graphically.

The following commands now potentially generate top and bottom sides of shell surfaces as output:

3. `*select_surfaces`,
   `*select_surfaces_nodes`,
   `*select_surfaces_edges`,
   `*select_surfaces_faces`,
   `*select_surfaces_points`,
   `*select_surfaces_trim_points`,
   `*select_surfaces_trim_curves`,
   `*select_surfaces_geometry`,
   `*select_surfaces_cbody`

The following commands now accept top and bottom sides of shell curves as input:

1. `*add_cavity_curves`, `*remove_cavity_curves`,
   `*add_srfprop_curves`, `*remove_srfprop_curves`,
   `*add_apply_curves`, `*remove_apply_curves`,
   `*select_curves` (SINGLE or ASSOCIATION method),
   `*store_curves`, `*remove_set_entries`

2. `*select_nodes_curves`,
   `*select_edges_curves`,
   `*select_points_curves`,
   `*select_surfaces_trim_curves`

A mixed list of non-shell curves and sides of shell curves may be given in these commands.

For shell curve `c1`, the top side is given by `c1:0` (or `c1`) and the bottom side by `c1:1`. A non-shell curve `c2` is given by `c2` (or `c2:0`).

Top or bottom sides of shell curves cannot yet be selected graphically.

The following commands now potentially generate top and bottom sides of shell curves as output:

**3.** `*select_curves`,
`*select_curves_nodes`,
`*select_curves_edges`,
`*select_curves_points`,
`*select_curves_geometry`,
`*select_curves_cbody`

The following commands now accept top and bottom sides of 3-D shell elements as input:

**1.** `*add_cavity_faces`, `*remove_cavity_faces`,
`*add_srfprop_faces`, `*remove_srfprop_faces`,
`*add_apply_faces`, `*remove_apply_faces`,
`*select_faces` (SINGLE or ASSOCIATION method),
`*store_faces`, `*remove_set_entries`,
`*select_elements` (PLANE method),
`*select_edges` (PLANE method),
`*select_faces` (PLANE method),
`*select_nodes` (PLANE method)

**2.** `*select_nodes_faces`,
`*select_elements_faces`,
`*select_edges_faces`,
`*select_surfaces_faces`

A mixed list of 3-D continuum element faces and 3-D shell element top and bottom faces may be given in these commands.

For 3-D shell element `e1`, the top face is given by `e1:0` and the bottom face is given by `e1:1`. Face `i` of a 3-D continuum element `e2` is given by `e2:i`.

Top or bottom faces of 3-D shell elements can be graphically selected if shell elements are drawn in expanded mode.

The following commands now potentially generate top and bottom faces of 3-D shell elements as output:

**3.** `*select_faces`,
`*select_faces_nodes`,
`*select_faces_elements`,
`*select_faces_edges`,
`*select_faces_surfaces`

The following commands now accept top and bottom sides of axisymmetric shell elements as input:

1. `*add_cavity_edges, *remove_cavity_edges,`
   `*add_srfprop_edges, *remove_srfprop_edges,`
   `*add_apply_edges, *remove_apply_edges,`
   `*select_edges` (SINGLE or ASSOCIATION method),
   `*store_edges`, `*remove_set_entries`

2. `*select_nodes_edges,`
   `*select_elements_edges,`
   `*select_faces_edges,`
   `*select_curves_edges,`
   `*select_surfaces_edges`

   A mixed list of axisymmetric continuum element edges and axisymmetric shell element top and bottom edges may be given in these commands. For axisymmetric shell element `e1`, the top edge is given by `e1:0` and the bottom edge is given by `e1:1`. Edge `i` of axisymmetric continuum element `e2` is given by `e2:i`.

   Top or bottom edges of axisymmetric shell elements can be graphically selected if shell elements are drawn in expanded mode.

The following commands now potentially generate top and bottom edges of axisymmetric shell elements as output:

3. `*select_edges,`
   `*select_edges_nodes,`
   `*select_edges_elements,`
   `*select_edges_faces,`
   `*select_edges_curves,`
   `*select_edges_surfaces`

### Boundary Conditions

In the menus for (`edge`) and (`face`) boundary condition types, the BOTTOM buttons and the text (TOP) have been taken out, because (`edge`) and (`face`) boundary conditions on the bottom of a shell must now be the defined using the first (former “top”) dof and applying it to the bottom side of a shell.

It was necessary to flip the direction of a positive pressure on edges of axisymmetric shell elements, as compared to all previous MSC.Marc Mentat versions. It is now identical to the MSC.Marc convention. Special care has been taken to deal with backward compatibility for this change.

The drawing of negative loads on edges (`faces`) has been changed. Now, the arrow points away from the object if the load has a negative magnitude.

### Backward Compatibility

Backward compatibility of model files (writing/reading old-style) and MSC.Marc input files (writing/reading old style (non-table-input)) has been achieved to a large extent.

Backward compatibility of procedure files (running 2003-style) can be strongly improved by inserting the command `*set_shells_old on`.

## 14. Menus

### Main

The order of the buttons in the MAIN menu has been changed. Assuming the user works through the menus from the top down to the bottom of the main menu, it has been ensured that wherever a user can reference a database entity, that entity has already been created.

For shell element modeling, it is essential that shell element types or shell geometric properties have been assigned before top or bottom edges (`faces`) can be selected.

### Modeling Tools

A new menu, MODELING TOOLS, can now be accessed from the MAIN menu. It contains gateways to the following submenus:

| | |
|---|---|
| TRANSFORMATIONS | (Also in MAIN→BOUNDARY CONDITIONS→ MECHANICAL) |
| CAVITIES | (Moved from MAIN→MESH GENERATION) |
| CROSS-SECTIONS | (Moved from MAIN→BOUNDARY CONDITIONS) |
| MATCHING BOUNDARIES | (New!) |
| FRACTURE MECHANICS: 2-D CRACKS | (Moved from MAIN→FRACTURE MECHANICS) |
| FRACTURE MECHANICS: 3-D CRACKS | (Moved from MAIN→FRACTURE MECHANICS) |
| WELD PATHS | (New!) |
| WELD FILLERS | (New!) |

### Mesh Generation

The room created by removing the CAVITIES button from the MESH GENERATION menu has been used to add a button leading to the ELEMENT TYPES menu.

### Surface Properties

A new menu, SURFACE PROPERTIES, can now be accessed from the MATERIAL PROPERTIES menu. It allows defining the EMISSIVITY for lists of edges (`faces`) instead of defining it for lists of elements in the HEAT TRANSFER material property type. This allows different emissivity values for edges of the same element, especially for the top and bottom face of a shell element. Note that SURFACE PROPERTIES can also be applied to curves (`surfaces`), in which case all edges (`faces`) attached to these curves (surfaces) will inherit the surface property.

### Boundary Conditions/initial Conditions

The boundary condition types and initial condition types that are considered to be "State Variables", have been moved to separate submenus of the BOUNDARY CONDITIONS and INITIAL CONDITIONS menus, named STATE VARIABLES.

For example, the thermal state variable types NODAL TEMPERATURE and STATE VARIABLE have been moved from the MECHANICAL submenu to the STATE VARIABLES submenu. The reason is that in case of temperature dependent properties, the option not to have temperature as a degree of freedom but to prescribe the temperature field is not restricted anymore to mechanical analyses, but can also be used in other analysis classes.

In addition, the AXISYMMETRIC TO 3D initial condition type has been moved to the new submenu INITIAL CONDITIONS→GENERAL, where one can also find two new initial condition types, PREVIOUS ANALYSIS STATE and 2D TO 3D.

The new boundary condition type GLOBAL-LOCAL can be found in the new submenu BOUNDARY CONDITIONS→GENERAL.

## 15. New-Style Radiation

Support has been added for the New-Style radiation analysis, which uses the new CAVITY RADIATION boundary condition type on cavities.

Viewfactor computation for a cavity using the HEMI-CUBE PROJECTION METHOD cannot yet be run from within MSC.Marc Mentat, but triggering the computation of cavity viewfactors during the MSC.Marc analysis is already supported.

The CAVITIES menu, and the JOBS→...→ANALYSIS OPTIONS→RADIATION menu have been enhanced.

## 16. New-Style Tables Input

All menu items needed for the support of the New-Style Tables Input are now visible. Previously, they could only be seen if MSC.Marc Mentat was run with the options `-df NEW_INPUT -mf main.ms`.

**Note:** The New-Style Tables Input is not yet on by default.

**Critical Notes:** If independent variable type used on a boundary condition, or contact is normalized time, increment number, normalized increment number or loadcase, you must use the NEW-STYLE TABLE input.

If an independent variable type is a coordinate to location; x, y, z, s, r, $x_0$, $y_0$, $z_0$, $s_0$, $r_0$, arc length, normalized arc length, distance from neutral axis, normalized distance from neutral axis is used, then you must use the NEW-STYLE TABLES input.

If a material data uses a table, and the independent variable is not temperature, equivalent plastic strain, or strain rate, you must use the NEW-STYLE TABLES input.

To activate the output of a Marc data file using the NEW STYLE TABLES driven input, use either the JOB→...→JOB PARAMETERS menu or on the JOB→RUN→ADVANCED JOB SUBMISSION menu.

## 17. Changed Defaults

1. In loadcases, the AUTO SWITCH option is now on by default.
2. It is no longer possible to deactivate iterative penetration contact procedure in the JOB→...→CONTACT CONTROL→ADVANCED CONTACT menu.
3. The EXTRAPOLATE option in the defining tables menu is now on by default for all independent variables of a table. For backward compatibility, upon reading of older model files, the extrapolate flag is always set if the independent variable is `temperature`, `eq_plastic_strain`, `eq_plastic_strain_rate`, `relative_density`, `magnetic_induction`, `eq_stress`, `eq_creep_strain` or `gasket_closure_distance`.

## 18. MSC.Marc Mentat 2005 Preprocessing Enhancements

1. The model or post file name now appears in the window title bar.
2. A command `intersect_extend_curves` has been added that extends two lines to their intersection.
   It is available using the MESH GENERATION→INTERSECT→EXTEND CURVES button. It will also truncate the curves if they cross, unlike the `intersect_curves` command available with the CURVE/CURVE button.
3. You can now add a circle by specifying three points. It is of type POINT/POINT/POINT, located in the MESH GENERATION→CURVE TYPE menu.
4. Python support has been upgraded from version 1.5.1 to version 2.2.2.
5. New database formula functions have been added. See the *MSC.Marc Python Tutorial and Reference Manual* for a complete list of functions.
6. An option to renumber imported models has been added. It is available by selecting the FILES→IMPORT→IMPORT RENUMBER button.
7. A button for CONSTANT TEMPERATURE has been added to the GEOMETRIC PROPERTIES→MECHANICAL menus for solid composite elements and gaskets.
8. The menus for FRICTION under JOBS→MECHANICAL→CONTACT CONTROL have been modified to reflect changes in the available friction solution methods.
9. Support for the FLUID DRAG loading factor has been added. A (time dependent) scale factor can be given for the fluid drag effect.
10. A command MERGE DUPLICATE INITIAL CONDS `[*merge_iconds]` has been added to the INITIAL CONDITIONS menu to enable merging duplicate initial conditions.
11. The list of table types displayed in the TABLE TYPES menu has been limited to only those types that are actually supported by the menus. Also, the list of selectable tables for SPRING properties and GASKET material properties has been limited to show only those tables that have an allowable type.
12. Two commands have been added to enable removing a list of RBE2's or RBE3's. Note that graphical selection is possible.

13 Several new automatic domain decomposition methods have been added. The DOMAIN DECOMPOSITION menu has been enhanced to support these methods. A roller button has been added to switch between display of manual decomposition menu items and display of automatic decomposition menu items. Also, a new command RENUMBER (DDM) `[*renumber_ddm]` has been added that renumbers nodes and elements if they are not yet consecutively numbered.

14 The option CONVECTIVE TERMS is now supported for JOULE analysis.

15 A toggle button TIME INTEGRATION ERROR CHECK has been added to support the Bergan check option for DYNAMIC TRANSIENT loadcases using the ADAPTIVE MULTI-CRITERIA stepping procedure (a.k.a. AUTO STEP).

16 A toggle button has been added in the ANALYSIS options menus to toggle the ROTATIONAL INERTIA TERMS for shell elements.

17 The button MAX TEMPERATURE CHANGE ALLOWED button has been moved from CONVERGENCE TESTING to ADAPTIVE:MULTICRITERIA PARAMETERS→DEFAULT CRITERIA and ADAPTIVE:TEMPERATURE PARAMETERS. Also, a button MAX VOLTAGE CHANGE ALLOWED has been added to the ADAPTIVE STEPPING PARAMETERS menus of the JOULE and JOULE-MECHANICAL loadcase types.

18 EXPERIMENTAL DATA FIT support for viscoelastic foam material has been added.

19 A better algorithm has been implemented for computing the position of mid-edge nodes of quadratic edges such that the edge is divided in two equal parts.

20 Support has been added for viscoelasticity for FOAM material.

21 Support has been added for user subroutine UELASTOMER for OGDEN and FOAM materials.

22 Help has been added for ELEMENT CLASS `[*set_element_class]`.

23 A menu has been added to set DDM SOLVER OPTIONS.

24 A command EDIT `[*md_table_edit_f]` has been added in the TABLES menus to allow editing the function value of a point in a multi-dimensional table. Note that the point to be edited can be graphically picked.

25 Buttons have been added to read ADDITIONAL INPUT FILE TEXT from external files.

26 Several enhancements have been made in the I-DEAS reader.

27 The ACIS libraries have been upgraded from version R5.2 to version R13. This provides support of the solids modeling capability to Linux and 64-bit platforms and fixes many previous defects associated with solid modeling.

28 When loads are expanded in a AXISYMMETRIC MODEL TO 3D EXPAND operation, all edge load types are now converted to their face load type counterparts. Previously, this was only done for the `edge_load` boundary condition type.

**Note:** Currently, the load expansion is only done for quad elements.

29 The PDE/Lib libraries have been upgraded to version 4.4.0. This provides many fixes to defects, primarily with the DXF and IGES translators.

The current versions supported by the import interfaces are:

| | |
|---|---|
| ABAQUS | 6.3 |
| ACIS | R13 |
| CMOLD | 98.7 to 99.1 |
| DXF | AutoCAD 2000 |
| I-DEAS | MS7 |
| IGES | 5.2 |
| NASTRAN | v70 |
| PATRAN | v8 |
| STL | Oct 1989 |
| VDAFS | 2.0 |

30 Python support has been enhanced with new functions and commands. The function `py_get_data`. similar to `py_get_float`, returns values from the database for materials, loadcases, contact bodies, contact tables, jobs, boundary conditions, initial conditions, geometry, local adaptivity, and global remeshing. See the description of the function in the *MSC.Marc Python Reference Manual*.

Also available is the ability to store data using the command *user_defined_data. It has the syntax:

`*user_defined_data` <index> <value>

<index> is a value in the range of 1 to 500.

The data may be saved to a text file using the command:

`*user_defined_Write` <filename> <yes>

The data may be read from a text file using the command:

`*user_defined_read` <filename>

A Python script may obtain this data using the `user_def (index)` function. It takes an integer value 'index' which is a number in the range of 1 to 500. It is described in the *MSC.Marc Python Reference Manual: Appendix A* under the Utility functions.

31 The thickness direction of solid composite and gasket elements can now be visualized via the SOLID COMP./GASKET PLOT SETTINGS menu. This menu can be accessed both from the various GEOMETRIC PROPERTIES menus and from the PLOT→ELEMENT SETTINGS menu. The thickness direction is indicated by an arrow that points from the bottom face of the solid composite or gasket element to the top face of the element.

Note that visualization of the thickness direction is available only if the elements are drawn in wireframe mode. Please refer to *Chapter 2.2* of the *MSC.Marc 2005 User's Guide* for an example.

32 An Abaqus reader has been added to the FILE→IMPORT menu.

33 Commands have been added to clear the selection of a single entity type only: `*select_clear_nodes`, `*select_clear_elements`, etc. The existing command `*select_clear` clears the selection for all entity types.

34 The support of the MSC.Marc functionality RELEASE NODE for gradual release of fixed displacements has been changed. The boundary condition type RELEASE NODES is no longer used for this purpose. Instead, it is now possible to select FIXED DISPLACEMENT boundary conditions as GRADUALLY RELEASED LOADS in LOADCASES→...→SELECT LOADS. Note that fixed displacements must have been selected as APPLIED LOADS in the loadcase prior to the loadcase in which they are selected as GRADUALLY RELEASED LOADS.

35 The ACOUSTIC BODY has been renamed to ZERO STIFFNESS BODY. Apart from an ACOUSTIC-SOLID analysis, it can now also be used in an ELECTROSTATIC-STRUCTURAL analysis.

36 Commands have been added to clear all text lines in ADDITIONAL INPUT FILE TEXT menus.

37 The local coordinate directions of coordinate systems defined at the nodes using the TRANSFORMATIONS menu can now be visualized via the TRANSFORMATION PLOT SETTINGS menu. This menu can be accessed both from the TRANSFORMATIONS menu and from the PLOT→NODE SETTINGS menu. If transformations are drawn, the local coordinate systems are indicated by three arrows, pointing from the node in the direction of the local x-, y- and z-axis. To distinguish the directions, the local x-direction is indicated by a red arrow, the local y-direction by a green arrow, and the local z-direction by a blue arrow. Plotting of each of these arrows can be controlled individually by the FIRST, SECOND, and THIRD DIRECTION buttons in the TRANSFORMATION PLOT SETTINGS menu.

Note that transformation will be displayed only if the nodes are drawn. Please refer to *Chapter 2.2* of the *MSC.Marc 2005 User's Guide* for an example.

38 A new command `*mesh_curves` has been added that automatically meshes a list of curves with line elements. The command is similar to the `*convert_curves` command in the MESH GENERATION→CONVERT menu, but uses the curve's divisions defined in the MESH GENERATION→AUTOMESH→CURVE DIVISIONS menu to generate the line elements, instead of a fixed number of divisions.

For each curve, the `*mesh_curves` command places nodes at the seed points of the curve and meshes each division with a single line element. If no divisions are applied, the curve is meshed with a single element. The command attaches the edges of the line elements to the curve and the nodes at the end points of the curve to these points. The new command can be found in the new MESH GENERATION→ AUTOMESH→ CURVE MESHING menu.

39 Because of the very different nature of 2-D and 3-D magnetostatic boundary conditions, three new types have been introduced especially for 3-D:

`fixed_3d_magn_potential` (ax, ay, az),
`magn_3d_point_current` (jx, jy, jz),
`magn_3d_volume_current` (jx, jy, jz).

The existing boundary condition types now only have one degree of freedom and are only used for 2-D and axisymmetric:

`fixed_magn_potential` (a),
`magn_point_current` (j),
`magn_volume_current` (j).

Backward compatibilty to/from older model files has been achieved. Older procedure files for magnetostatic analyses may have to be edited because of changed bc type names or dof names.

40 In electrostatic and magnetostatic jobs, boundary conditions cannot be selected as initial loads from now on. The loads for the steady state computation must be selected in the loadcase. Backward compatibility of older model files and procedure files for this change has not yet been achieved, so the user must check if the desired loads are active in the loadcase.

41 The LOADCASE RESULTS menu button has been moved to the bottom, and now give access to an enhanced LOADCASE RESULTS menu in case MULTI-CRITERIA stepping has been switched on. The RESULTS AT FIXED TIME INTERVALS functionality has been moved to this enhanced menu.

## 19. MSC.Marc Mentat 2005 Postprocessing Enhancements

1 The deformation scaling factor may now be displayed in the graphics window. It is enabled with the RESULTS→DEFORMED SHAPE→SETTINGS→SHOW FACTOR button.

2 An option to set the angle that determines a boundary outline for FIND SOLID OUTLINE (the `find_solid_outline` command) has been added. It is available in the RESULTS→DEFORMED SHAPE→SETTINGS menu.

3 A toggle button USER VECTORS has been added to toggle the recognition of user defined vectors in postprocessing.

4 Support has been added for reading large post files (> 2GB) on Windows 2000/XP.

5 Support has been added for the element scalar quantity Rebar Angle (post code 487).

# III. List of Defects Fixed in this Release

## MSC.Marc 2005

### ADAMS MNF

1 MNF file was wrong for models that contained nodes with local coordinate systems via the TRANSFORMATION option.

### Adaptive Meshing and Rezoning:

1 Avoid potential premature exit or wrong results if different element types are used in local adaptivity. This could for instance happen if elements 11 and 155 were both used in the job.

2 Avoid potential premature exit in a parallel run with local adaptivity due to inactive elements.

3 Improve data mapping for Herrmann elements with local adaptivity. This fix will not affect the end results, but the convergence after element subdivisions will be improved.

4 Add support for local adaptivity with elements 155 and 156.

5 The check for allowed element types in global remeshing was sometimes missing a disallowed element type. This has now been corrected.

6 Correct a problem with local adaptivity with tetrahedral elements with face loads. The face load was applied incorrectly for split elements.

7 Correct a problem with load controlled rigid contact bodies after remeshing. The control node associated with the load controlled body was not taken into account in the convergence testing after remeshing.

8 In 2-D remeshing with contact, the contact was sometimes lost directly after remeshing. This was due to an inaccurate projection to NURBS curves and has now been corrected.

9 If elements have edge or face sets in a job where local adaptivity is used, the post file could be wrong or the job could exit prematurely. This has now been corrected.

10 When using global remeshing and only an immediate remeshing was requested, the job could stop with an exit number 13. This has now been corrected.

11 When using global remeshing for a contact body other than the first contact body, the first estimation of the friction behavior right after remeshing may be inaccurate.

12 Nodal values were incorrectly mapped in a heat transfer analysis when a new mesh was created in remeshing. This has now been corrected.

**Contact and Tyings:**

1 Correctly perform check on the order of the retained nodes for refinement tying 33 if elements with different types are used. This tying is used for 8-noded brick elements, and the problem could occur if the model also contains elements with less number of nodes (e.g. 4-noded shell elements).

2 During the iterative penetration checking procedure, the thickness of shells was not taken into account for nodes of shell elements which were potentially touching an analytical deformable body.

3 Fix for motion of position controlled rigid bodies using older input format. If a target position for the center of rotation of a position controlled body was specified under the CONTACT data block of pre MSC.Marc2003 input files, the body would not move.

4 In a parallel analysis with deformable contact, penetrations could occur during sliding if a contact body is split between different domains. This has now been corrected.

5 Correction to the rbe2 tying for the case that it is connected to a solid element and a spring with a rotational stiffness.

6 Avoid rare cases of divide by zero when friction is used in beam to beam contact.

7 Fix potential errors in contact detection with iterative penetration checking in 3-D if there are NURBS in the model which contain spans of zero area.

8 Fix problems with iterative penetration checking in 2-D if a node touches two bodies in the same direction.

9 Fix error in contact boundary conditions during harmonic and modal loadcases. The boundary conditions from contact were lost in this case.

10 In rare cases with iterative penetration checking, a divide by zero could occur, causing an abnormal termination of the job.

11 In rare cases with 3-D contact where boundary conditions are used in the area of contact, the contact search could fail leading to contact penetration. This has now been corrected.

12 A special case in beam-to-beam contact was not treated correctly. If two contacting beam elements rotate while being in contact such that they end up lying in one plane, the results may be wrong.

13 In 2-D contact, symmetry bodies are automatically extended infinitely. This caused problems if the symmetry line was created with multiple segments with different directions. The automatic extension is now only done if the symmetry line consists of one segment.

14 Fix for the delayed slide option in 2-D. In some cases, it would not find initial contact in the slide off area if a nonzero bias factor was used.

15 In rare cases, the recycling due to contact would never converge. This has now been corrected.

16 In a parallel analysis, the boundary description of a contact body consisting of line elements could be wrong.

17 If in a 3-D analysis a node is touching 3 bodies, separation from the third body would not be possible. This has now been corrected.

18 In a multi-body contact analysis where the order in which the search for contact is done is either user-defined or set by the program based on the element dimensions, potential self-contact could worsen the final set of contact tyings.

19 Rigid body rotation of shells with glued contact would introduce artificial stresses when the shell thickness is taken into account in the contact. This has now been corrected.

20 The limit of maximum 80 holes in a 2-D contact body has been removed.

21 The combination of nodal transformations with RBE2 or condensed elements (Herrmann, element 157, piezoelectricity) would lead to incorrect results. This has now been corrected.

22 Linearized buckling with glued contact could in some cases produce incorrect results. This has now been corrected.

23 If the approach option was used with moving bodies that are not active in the contact table, the job would stop with exit 40. This has now been corrected.

24 If a higher-order element touches a lower-order element in 2-D heat transfer or friction, the results may be incorrect. This has now been corrected.

25 There are situations where, models with RBE2 subjected with large rotation, the motion of the tied node drifts away from the “exact” location. This is normally happening when only residual forces are set as convergence criteria with default values. It has been fixed in version 2005.

### Coupled Analysis

1 The convection to the environment for elements partly in contact in a thermomechanically couple analysis has been corrected.

### Element Formulation:

1 The load per unit length for two-dimensional elements has been corrected.

2 Corrections to the calculation of interlaminar shear for composite shells. Wrong components of generalized stiffness were being used while calculating the shear components `tau_23` and `tau_31`.

3 Added model error checking for the case that the beam direction is specified equal to the first local beam axis.

4 Correction to the orientation option for continuum elements used in the updated Lagrange framework. The total Lagrange case was not affected.

5 Support fluid drag loads for beam elements 9, 45, and 64. This was mentioned in the documentation but was not working.

6 Correction for the “point temp” option for elements 155, 156, and 157. The bug had the effect that wrong integration point temperatures occurred.

**Material Models:**

1 Fix problems with the soil model:
- Correction to the formulation of soil analysis with pore pressure. The results for this case were incorrect.
- Correction to the linear soils model when it is used together with the ELASTIC option.
- Correct a problem related to fluid gravity load. The input related to the SPECIFIC WEIGHT option was interpreted incorrectly.

2 Correct the TRIP strain calculation for shape memory alloy models.

3 Fix for thermal expansion of rubber materials in the updated Lagrange framework. The results would be incorrect in this case.

4 The combination of thermal expansion for viscoelasticity with the Mooney material model in the total Lagrange framework was giving incorrect results. This has now been corrected.

5 Force a reassembly of the stiffness matrix if there are tables for gaskets that depend on temperature.

6 Correction to the ORIENTATION option for orthotropic material in a 3-D magnetostatic analysis.

7 When the kinematic hardening plasticity model is used with shell elements, the job could in rare occasions exit prematurely. This has now been corrected.

8 Correction to the use of the user subroutine HYPELA for hypoelastic materials. Routines that worked in versions prior to MSC.Marc 2003 may not work. This is now fixed so that it works as before.

9 Corrections for implicit creep option:

(a) secant tangent was not calculated correctly when few elements were being used for strain/stress check in auto creep

(b) stress change due to temperature dependent elastic properties were not incorporated correctly for materials with creep

(c) radial return tangent was not calculated correctly when UCRPLW was used to define the creep coefficients.

10 In a magnetostatic analysis, the check on the validity of the permeabilities for an orthotropic material did not take into account the definition via the B-H relation option. This could erroneously result in exit number 13.

**Memory Management**

1 The problem with memory reallocation on the Microsoft Windows platform has been fixed. With previous versions, an increase in memory allocation where the sum of the old and new memory exceeded GB would fail.

2 A large number of set definitions could lead to an exit 6 so the job would not run. This has now been corrected.

**Miscellaneous:**

1 Properly update time step after local adaptivity for fast explicit dynamics. The stable time step update was lagging by one increment for adaptive meshing jobs using fast central difference.

2 Correction for the case of multiple element types with the cylindrical option when elements have different number of degrees of freedom.

3 Correction to certain cases where the cylindrical option is used. Rubbish results values could appear in increment 0 (when it was a null step) which in some cases could lead to that the job stopped abruptly.

4 Avoid problems with the rebar verification files. Conflicts with the control file and spline verification file could occur.

5 Make sure the rebar verification file is opened in the working directory. Previously, it was opened in the directory where the input files are. These are not always the same.

6 Improvements in fluid calculations:
- Correct strain rate and stress calculation (postprocessing only).
- Allow film conditions to be used in a coupled fluid-thermal analysis.

7 Fix the axito3d option of using the last increment from the axisymmetric run in a parallel run.

8 Avoid problems when an element is inactive at the start of the job and activated later. This is for the case of multiplicative plasticity.

9 Avoid potential problems in jobs with harmonics. In rare occasions, the job would exit prematurely.

10 Avoid potential problems (premature exit or wrong results) with POINT CURRENT in magnetostatics or electromagnets and POINT CHARGE in electrostatics.

11 For a parallel job, the wrong node was reported to have the maximum error in temperature estimate (the results were correct, though).

12 When user subroutine CRPLAW is used, swelling strains could only be used if the equivalent creep strain increment was defined, but not if the full incremental creep strain tensor was defined. This has now been corrected.

13 Allow the sparse iterative solver to be used with steady state rolling.

14 In the metal cutting feature, the angles in the cutter definitions were not properly converted from degrees to radians.

15 Avoid premature exit of a job with the metal cutting feature. This could happen in cases where the cutting process takes less increments than the total number of increments specified.

16 Fix problems with the ELSTO option. This includes fixes for
- Analysis with composite brick elements.
- Superplastic forming.
- Certain cases of local adaptivity.

17 Correct printout of timing information so it prints something also for very long runs.

18 Correct format for printout of memory summary for solver. For parallel runs the total memory could go beyond 8 GB and then it was printed as "*****".

19 Correction for the case that springs are used in an analysis with brick-to-shell transitions and the last element in the model is a brick element with a brick-to-shell transition. This case would lead to incorrect spring forces.

20 Some printout of 2-D NURBS data was not suppressed when the option to suppress this info was turned on.

21 Wrong results could occur in certain cases of fluid solid coupled analysis. This has now been corrected.

22 Correction to cyclic symmetry for the case that the point on the symmetry axis is not at the origin. For this case, the results would be wrong.

23 Fix problem in axito3d option when used with rebar elements. In certain cases, the orientation of the rebar could be incorrect in the 3-D job.

24 Improvements to transient dynamics using the auto step option. The time step calculations are now less sensitive to velocity overshoot issues associated with Single Step Houbolt (SSH). Also, the transitioning of the time step from one dynamic loadcase to the next is more smooth. Finally, a smaller default time step stepping factor is provided for SSH (to reduce amount of possible artificial damping) with the option that the user can change it if needed (previously the factor was hard-wired).

25 With user subroutine GENSTR, the strain energy of the element was not calculated. This has now been corrected.

26 If the memory needed for element data is larger than 8 GB and the ELSTO option is not used, the job would stop prematurely due to an integer overflow. This has now been corrected such that the ELSTO option is automatically turned on in this case.

27 In a linear transient dynamic analysis, the control flag to force the assembly of the operator matrix every recycle did not work properly. This has now been corrected.

28 Correction to the auto step option with artificial damping using Herrmann elements. The pressure degree of freedom should not have a mass.

29 Correction to the options ACC CHANGE and VELOCITY CHANGE for dynamics when using the AUTO STEP option.

30 Avoid possible premature exit in the Lorenzi J-integral calculation if more elements than what is actually used is specified with the parameter “sizing”.

31 Fix problems with the Lorenzi J-integral:
- 2-D Herrmann elements with multiple crack tip nodes.
- Two-way degenerated element at crack front.
- Shift vector if only one element layer at crack front.

32 The combination of follower force and foundation could in some cases lead to abnormal termination or incorrect results. This has now been corrected.

33 The INSERT option would in some case give incorrect results in a thermomechanically coupled analysis. This has now been corrected.

**Parallel Processing:**

1 Depending on the machine configuration, a parallel network job could in some cases not be started from `c:\temp` on the Microsoft Windows platform. This has now been corrected.

2 Avoid potential problems with sharenames on Microsoft Windows for network jobs.

3 In certain cases of contact with separation in a parallel run the job could stop abruptly. This would only happen if the flag in the contact option for maximum number of separations in an increment was set to a small number.
By default, it is 9999. This problem has now been corrected.

4 Added a utility routine “domnodmask” which is referenced in *MSC.Marc Volume D: User Subroutines and Special Routines* for parallel processing but was missing in the code.

5 When the RESTART option to have one restart file for each saved increment is used, there was no base file created. This base file is not used in the run, but it is used from MSC.Marc Mentat and MSC.Patran when selecting which job to restart from. This file is now created properly.

6 When using the user subroutine FORCDT for specifying point forces and point fluxes, the loads applied to nodes shared between domains would be applied multiple times. This has now been corrected.

7 Two separate cases of restart in a parallel run would fail:
- Sparse iterative solver with the incomplete Cholesky preconditioner.
- Local adaptivity.
This has now been corrected.

**Preprocessing, Postprocessing, and Output:**

1 Avoid premature exit or rubbish on post file if post codes with quantities that are not used in the analysis are specified, for instance, creep strains in a non-creep analysis.

2 Avoid printing rubbish values to the output file for quantities not present in the analysis, for instance, plastic strains if no plasticity is present in the model.

3 The post file would contain wrong results for the case that there are no elements in the model. This could be the case, e.g. if only springs and/or mass points are defined. This has now been corrected.

4 Fix problems with the post file for shape optimization and design sensitivity. The post file was incorrect and could not be used. With a post file format of version 2000 and earlier, it would be correct. This problem is now fixed.

5 Correct the creep strains given in the output file using the PRINT NODE option.

6 Fix the writing of the Hypermesh file.

7 Avoid potential problems during the reading of input files for large models.

8 Avoid problems when comments using the character “$” are used in certain places.

9 Avoid printing rubbish values to the post file if post codes 108 and 109 are used in a model with composite solid elements. These post codes are for interlaminar shears for beam and shell composites and should be printed as zero for solid composites.

10 Avoid problems for the case that a boundary condition name is a subset of a previously defined boundary condition name.

11 Allow the generation of flow line grids for element types 155, 156, and 157.

12 Allow the input of the print element option to have an empty line for the layer number. Previously, this would give an error.

13 Allow an invalid degree of freedom to be specified in the input for boundary conditions. Now instead of an error, a warning is given and the job will continue with the invalid degree of freedom filtered out.

14 Allow the deformed rebar angle to be put on the post file and be written to the output file.

15 Fix the reading of potential change in a magnetostatic analysis.

16 Allow access to potential, external, and reaction charges in piezoelectric analyses via user utility routine NODVAR.

17 Fix postprocessing of contacted body for thermal contact.

18 Correct sign of reaction charge on the post file for a piezoelectric analysis.

19 Intermediate results during iterations on the post file were wrong for the case of a moving heat transfer rigid body.

20 The suppression of echo of certain input with the no echo option did not cover points and faces. This has now been corrected.

21 Correction to problem with the summary option for magnetostatics.

22 The user subroutine PLOTV did not have the correct argument values for non-structural problems (thermal, magnetostatics, electrostatics). This has now been corrected.

23 Fix a problem when reading DIST CURRENT in the history definition part in Joule heating analysis. In certain cases, one would get an exit 13. This has now been corrected.

24 The post file was incorrect for an electrostatic analysis with shell elements. This could cause MSC.Marc Mentat to crash. This has now been corrected.

25 Complex user defines nodal variables via user subroutine UPSTNO were wrong on the post file. This has now been corrected.

**Solver:**

1 Fixed bug in out-of-core for solver 8 in parallel.

2 A job using solver 4 in out-of-core mode in a dynamics job without tyings could in some cases exit prematurely. This has now been fixed.

3 Fix problem with out-of-core solver 8. In rare occasions, one would get an exit number 2006.

4 Fix problem with out-of-core solver (all solvers) for dynamics with damping. The results would be incorrect in this case.

5 Fix problem with out-of-core solver (all solvers) for fluid-solid coupling on the Microsoft Windows platform. In some cases, the results would come out wrong. This has now been corrected.

6 The performance of solver 8 (multifrontal) out-of-core was bad on Microsoft Windows, Compaq/HP Alpha, and SGI. Direct access scratch files were four times larger than on other platforms due to a different definition of record length. This has now been corrected and the scratch files are of the same size as on other platforms.

Also, several changes have been made to documentation and data files in the demo directory.

## MSC.Marc Mentat 2005

**Preprocessing:**

1. Picking edges of a quadratic element when PICKING mode is COMPLETE has been corrected.
2. Changing plot settings for an RBE2 without tied nodes now works correctly.
3. The energy function of Gent material model is now computed correctly.
4. The number of points needed for a grid is now calculated correctly. It used to cause a problem with the node spacing being incorrectly drawn.
5. Sweeping of nodes belonging to RBE3's is now possible.
6. Nodes that are shared by elements and RBE2's or RBE3's are now drawn if the elements are made invisible.
7. On some platforms (Sun, Digital Unix, Linux), the MSC.Marc Mentat program can only be run if certain shared libraries were installed. This is no longer needed.
8. IDENTIFY BACKFACES `[*identify_backfaces]` was incorrect if a sphere or cone surface with trimming curves was drawn in SOLID mode without LINES.
9. The command MERGE DUPLICATE BOUNDARY CONDS `[*merge_applies]` could fail to merge boundary conditions if a boundary condition had been imported from a pre-2003 model file.
10. The command APPLY `[*xcv_apply]` used in an experimental data fit for damage models worked inconsistently. If the current material was an OGDEN material, it would change it to MOONEY; if the current material was not an OGDEN material, it would change it to OGDEN.
11. The commands BETWEEN NODE `[*add_node_between]` and BETWEEN POINT `[*add_point_between]` were incorrect if the user coordinate system was CYLINDRICAL or SPHERICAL and the phi-coordinate of the first point was less than -90 and the phi-coordinate of the second point was greater than 90 or vice versa.
12. A problem with SUBDIVIDE of element faces attached to closed or collapsed surfaces has been corrected. Faces that crossed the boundary where the surface meets itself were subdivided wrongly (nodes were placed on the wrong part of the surface).
13. A problem existed with ATTACH of an element edge to a global curve, in case one of the faces that share the edge was already attached to a surface, and the global curve would lie entirely in that surface.
14. The command APPLY CURVE DIVISIONS `[*apply_curve_divisions]` is now protected against a division by zero error if an empty curve list is given. The same problem existed for the obsolete command `*curve_length`.
15. When elements are subdivided that have edges or faces in a cavity, the new elements would not have the corresponding edges or faces in the cavity. The same problem existed for CHANGE CLASS and refine.
16. Elements that have edges or faces in a cavity could not be removed.

17 New elements created by the SUBDIVIDE commands ELEMENTS TO QUAD [*subdivide_elements2quad] and ELEMENTS TO HEX [*subdivide_elements2hex] now inherit the material property, geometric property, contact body, orientation and set membership of the original element.

18 It was not possible to remove a table if it was previously referenced by a parameter with a value different from the default value. The problem only existed for boundary condition, contact body, contact table, and spring parameters.

19 The input data checking for GLOBAL REMESHING criteria has been improved.

20 If a mesh of 3-D continuum elements contained collapsed hexahedral elements, the program generated an incorrect list of faces on the SURFACE of the mesh. The same problem existed for the list of edges on the OUTLINE of the mesh.

21 An error is now given when a COUPLED job doesn't have thermal material data, a JOULE-MECHANICAL job doesn't have joule material data, or a FLUID-THERMAL/FLUID-THERMAL-SOLID job doesn't have thermal material data.

22 The EXPERIMENTAL DATA FIT related to the biaxial deformation mode of MOONEY materials was incorrect.

23 The command line option -xfdb has been fixed. Both -xfdb false and -xfdb true would switch off fast double buffering.

24 ALIGN SHELLS [*align_shells] did not work for quadratic shell elements.

25 For a REBAR material with the LAYER TYPE set to SKEW, DUPLICATE LAYER [*duplicate_layered_material] did not correctly copy the parameters REL. LAYER POSITION, REBAR AREA, # REBARS / LENGTH, and ANGLE. Moreover, the commands CLEAR LAYERS [*clear_layered_material], DUPLICATE LAYER, and COPY LAYERS [*copy_layered_materials] did not set the THICKNESS DIRECTION correctly.

26 The command COPY TO CLIPBOARD [*copy_to_clipboard] has been fixed, so it can deal with an empty title string.

27 The command to READ a table [*md_table_read] could not read table files written in the pre-2003 format.

28 CHANGE CLASS would be incorrect if faces were attached to a surface whose boundary at v=1 was collapsed to a point.

29 A problem with SYMMETRY COMBINED [*symmetry_combined] has been fixed. Attach information between edges and trimming curves could be wrong for the copies. This only happened if *set_symmetry_geometry_old was off (the default).

30 A problem with SUBDIVIDE [*subdivide_elements] for TRIA3 and TRIA6 elements has been fixed. If (face 0 of) the original element was attached to a surface, then the program failed to attach (face 0) of all the new elements to that surface.

31 A problem with CHANGE CLASS has been fixed. Surfaces with collapsed boundaries at v=0 or v=1 were not recognized.

32 It was impossible to make a scalar plot of components of element integration point vectors in a non-default RESULTS COORDINATE SYSTEM.

33 Copying an RBE3 that has a retained node would cause the program to crash.

34 Copying a contact body that has a control node assigned to it might lead to problems.

35 The program would give warning messages if EDGE RADIATION or FACE RADIATION boundary conditions were selected in a loadcase or job.

36 A problem has been fixed in the Marc reader with respect to reading geometry attach information and BEAM SECT properties.

37 In a DDM contact job, the maximum number of nodes on the boundary of a contact body was computed without checking if the nodes were part of the domain. As a result, the amount of memory allocated in MSC.Marc would be to high.

38 Surface lines would disappear when doing a SELECT `[*select_surfaces]` and MAKE VISIBLE `[*make_visible]` on a surface while in solid mode.

39 The MOVE COMBINED `[*move_combined]` command was incorrect if a node was attached to two or more surfaces. The same problem occurred for MOVE SURFACES `[*move_surfaces]`.

40 A small gap has been made between the CLEAR SELECT and MAKE VISIBLE buttons to prevent “accidents” by wrong clicks.

41 Model files with a nonzero number of hypermesh element variables could not be read.

42 Boundary conditions on cavities were not drawn, in case the cavity had been defined using curves/surfaces.

43 The `check_upside_down` command now first checks that the elements selected lie parallel to the XY plane, which is a requirement for this check.

44 An error is reported if a node transformation is not correct for 2-D and axisymmetric analyses.

### Postprocessing

1 The command CLEAN FILES `[*clean_animation, *clean_mpeg_animation]` in the ANIMATION menus now works when the index is unequal to the default of 100.

2 The plotting of integration point results for rebar element types 23, 46, 47, 48, 142 to 148, 165 to 170 and composite continuum element types 149 to 154 and 175 to 180 has been fixed. The average value would be shown.

3 The minimum value for beam plots was incorrect.

4 The contact body type was not set correctly when reading a post file produced by running an input file not written by MSC.Marc Mentat.

5 The LINEAR and TRANSLATE methods for EXTRAPOLATION of element results were wrong for element type 157. The AVERAGE method was used instead.

6 The TRANSLATE method for EXTRAPOLATION of element results did not work for TRIA6 and TETRA10 elements. It displays the same as the LINEAR method.

7 The reading of duplicate spring id's in a post file is now handled properly.

8 Scalar and tensor post results values now use different naming schemes. The tensor quantity computed is now listed as "Equivalent of". For example, the tensor quantity of Total Strain is now listed as "Equivalent of Total Strain".

9 The display of CONTOUR BANDS, CONTOUR LINES, and CONTOUR CENT plots are now displayed correctly for QUAD8, HEX20, or PENTA15 elements when using EXTRAPOLATION METHOD TRANSLATE.

10 The collection of subincrement data for HISTORY PLOT has been made more robust. If the begin increment is given by the user as `Inc` or `Inc:0` and the end increment as `Inc:n` with `n > 0`, the begin increment is corrected to `Inc:1`.

11 The reading of complex user defined nodal quantities from a post file, when given as Real and Imaginary part, has been corrected.

12 Vector plots and Scalar plots of vector magnitude for 3-D vectors in a 2-D analysis are now correct. This is seen in 2-D electromagnetic analysis, where Magnetic Potential is a 3-D vector.

### MSC.Marc Input File Writer

1 If a Marc input file is written using EXTENDED PRECISION (default), real numbers that have an absolute value in the interval [9.9999995e-10, 1e-9) are now written with a correct (zero) exponent.

If a Marc input file is written without EXTENDED PRECISION, real numbers that lie have an absolute value in the interval [9.999995e+9, 9.9999995e+9) are now written with a correct (zero) exponent.

2 An incorrect input file would be created if a loadcase with a RELEASE NODES boundary condition was preceded by a loadcase at the end of which a FIXED DISPLACEMENT boundary condition had a nonzero magnitude.

3 If a model already contains a table and a material is read `[*material_read]`, an input file written in the new style would be incorrect because of duplicate table ids.

4 An incorrect input file was written for FLUID-THERMAL-SOLID jobs if the option CONVECTIVE TERMS was on `[*job_option convective:on]`. As there is no button to set this option for this job class, this problem could only occur if the job class had been changed from e.g. HEAT TRANSFER.

5 An incorrect input file would be produced if the # STEPS is smaller than 1 for a STATE VARIABLE boundary condition using the POST FILE method.

6 The writing of the SETNAME parameter option has been improved. The number of sets and/or set items was sometimes too high.

7 For a LINEAR ELASTIC analysis, an incorrect input file might have been written if a loadcase was selected with a # STEPS larger than 1. The same was true for DESIGN SENSITIVITY and DESIGN OPTIMIZATION analyses.

8 STATE VARIABLE boundary (or initial) conditions with a STATE VARIABLE ID unequal to 1 were not written to the Marc input file if they were selected in a job with a thermal pass.

9 An incorrect Marc input file was written when initial conditions INITIAL VOID RATIO, INITIAL PORE PRESSURE, or INITIAL POROSITY were active.

10 An incorrect input file was written if a model contained CAVITY PRESSURE LOAD or CAVITY MASS LOAD boundary conditions using the UCAV user subroutine.

11 When a model contained a loadcase, in which a NODAL TEMPERATURE boundary condition was deactivated and another NODAL TEMPERATURE boundary condition was activated, an incorrect input file would be created, if the two boundary conditions had nodes in common. Note that the POST FILE method implies that the boundary condition is applied to all nodes of the model.

12 If a model contained composite continuum or gasket elements to which a STATE VARIABLE boundary condition was applied using the ENTERED VALUES method, an incorrect Marc input file was written. The line specifying the list of integration points was not written. The same problem occurred for initial conditions applied to these element types.

13 For piezo-electric analysis, the point char option is now properly written to a dat file. It previously wrote the point cur option.

14 The computation of the maximum number of nodes on the boundary of a deformable body is now done with respect to the elements that are active in increment 0.

15 A correct input file is now written if multiple point loads (fluxes, charges...) are applied to the same node and one of these loads is not selected as initial load.

16 When results are requested for ALL or OUTER & MIDPLANE layers of 3-D beam elements (under JOB RESULTS), this information is now written to the file correctly. Previously, only layer 1 is was written to the post section.

17 The options THICKNESS and THICKNS CHANGE for bearing analysis are now supported.

18 If a model contains contact bodies that have nodes in common, a correct Marc input file is now produced.

19 If a model contains multiple transformations that uses the user subroutine UTRANS, a correct input file is now written.

### I-DEAS Reader

1 Several problems in the I-DEAS reader have been fixed.

2 A problem in the I-DEAS reader for quadrilateral membrane elements has been fixed.

# IV. List of Known Problems in this Release

## MSC.Marc 2005

**1.** There are several limitations to the table driven input format capabilities:

(1) Table input is not available with Fourier Analysis.

(2) Table input is not available with Pressure Cavity Loading.

(3) The definition of the table cannot change upon restart.

(4) Table input is not supported with element types 4, 8, 24, 51, 95, and 96.

(5) Table input does not support edge loads on shells.

(6) Table input is not available with Hydrodynamic Bearing.

(7) Table input is not available with Fluid analysis.

**2.** The following problems currently exist with the beam-to-beam contact:

– if there is contact between beam elements and nodes of these beam elements are also contacting solid or shell elements, the friction due to contact of solid or shell elements is not correct

**3.** The thickness of beam elements is currently ignored if (the nodes of) beam elements touch a rigid surface, a shell element, or a face of a solid element.

**4.** For remeshing with self-contact may show unwarranted penetrations. The use of *Optimize Contact Constraints* mentioned under *iv. Deformable Contact* subsection of *A. Contact Enhancements* in *1. Analysis Robustness* topic of the "Description of the New Functionalities" certainly alleviates the problem with penetration in self contact but does not completely remove it in all circumstances. This is still under development.

**5.** Mass lumping should not be done for element types 52 and 98 in explicit dynamic analysis using DYNAMIC,5 parameter.

**6.** Convergence of steady state rolling analysis using torque control is slower than when using spinning velocity control. It is useful to start with spinning velocity controlled solution and use torque control when the solution is close to the final one.

**7.** Thickness calculations for the plane stress elements, including shells, is based on incompressibility. This would not be correct for ISOTROPIC option where elastic strains are large in comparison with the total strains (the thickness strains are exact for rubber materials). This also yields incorrect thickness strains when using UPDATE, LARGE DISP, FINITE for plane stress elements in thermal analysis.

**8.** Pipe element does not account for gravity loads.

**9.** Harmonic analysis with point dampers produce incorrect reactions.

**10.** Element 22 gives incorrect buckling results.

**11.** Environment variables for parallel jobs (DDM) on the Microsoft Windows platform must be set using the System applet from the Control Panel. This means that if the variable MSC_LICENSE_FILE is to be set to point to a different license file, it cannot be changed by modifying the *include.bat* script or by setting it in your command prompt window.

**12.** In the non-table format, if a negative film coefficient is provided for a convective film boundary condition in heat transfer analysis or a negative foundation stiffness is provided for a foundation boundary condition in mechanical analysis, the solution obtained by MSC.Marc is not correct. Two workarounds are possible:

a. use the new table format to write out the input deck

b. Provide the negative coefficients through user subroutines FILM and USPRNG respectively.

## MSC.Marc Mentat 2005

1. **MSC.Marc import does not support element types 4, 8, 13, 15, 16, 17, and 24**

   Element types 4, 8, 13, 15, 16, 17, and 24 are not supported by the MSC.Marc reader (IMPORT MARC).

2. **Running MSC.Marc Job in Different Directory**

   If a model has been created by opening an existing model file, the MSC.Marc job is always run in the directory containing the model file.

3. **Drawing of symbols on Digital Unix very slow**

   The drawing of symbols (nodes, points, etc.) in the OpenGL version of MSC.Marc Mentat on Compaq/Digital Unix is very slow. This is due to a problem in the glBitmap implementation. This problem is seen when a model is first drawn with nodes or points turned on.

4. **MSC.Marc import and export does not support edge foundation on 3-D beam elements**

   For 3-D beam elements, the edge foundation option (BOUNDARY CONDITIONS→ MECHANICAL→EDGE FOUNDATION) is not supported by the MSC.Marc reader (READ MARC) or MSC.Marc writer (WRITE MARC).

5. **Recover option always requests both modal stresses and reactions**

   When doing an eigenvalue extraction, MSC.Marc Mentat always generates the RECOVER option to request both modal stresses and reactions. This can lead to slow eigenvalue analysis if these items are not needed. To disable this, specify a 0 in the 3rd field (columns 11 – 15) of the second data block of the RECOVER option if only the eigenvectors are required.

6. **Contact Status may show non-integer values in Local Adaptivity analysis**

   When postprocessing the scalar Contact Status in a Local Adaptivity analysis, some nodes may appear in red (a value of .5) when they should have an integer value of 0, so the nodes with a value of 0.5 are not in contact.

7. **Converting Solid Faces to Surfaces**

   Running the SOLID FACES TO SURFACES command under the SOLIDS menu will fail if there are any inside out faces. If the command fails, the solid should first be checked with CHECK ENTITIES before running this command. Run the CHECK ENTITIES command, and if any error is found, do not run the SOLID FACES TO SURFACES command.

8. **MSC.Marc Python feature availability**

   The external modules `py_mentat` and `py_post` are not available for IBM AIX.

9. **MSC.Marc Mentat OpenGL version on IBM**

   The OpenGL version of MSC.Marc Mentat may not operate properly on some IBM platforms. The runtime libraries patch 4.3.3.25 is required.

**10. MSC.Marc Mentat OpenGL version on Linux**

MesaGL v3.4 or higher is required to run the OpenGL version on Linux.

**11. Shell Expand for Line Elements**

If the element orientation of 2 connected elements is not the same (e.q. Connectivity of element 1: 1 2, of element 2: 3 2) the shell expand for line elements gives wrong quad elements.

**12. Fill View is Incorrect when not all Table Points are Visible**

If a table is created and then the XMIN, XMAX, YMIN, or YMAX value is specified such that not all of the data points will be visible, then when a FILL (`*fill_view`) command is specified the result will be incorrect.

**13. Models with large element id's may not post process correctly**

If a model has element id's which are very large and some very small, MSC.Marc Mentat will not post process the post file correctly. For example, if some element numbers are in the 100's and others are in the 75,000,000's, MSC.Marc Mentat will not handle this case properly.

**14. Translators for 64-bit MSC.Marc Mentat HPUX 11.0**

The translators `iges`, `dxf/dwg`, and `vdafs` are not available in 64-bit mode for MSC.Marc Mentat on HPUX 11.0 and Linux on Intel Itanium 2. They are the 32-bit builds of the translators.

**15. Saved view cannot be loaded on Sun Solaris and HP OpenGL version**

When a view is saved (SAVE VIEW) from the VIEW menu, it won't be read properly when using LOAD VIEW on the Sun Solaris and HP platforms with the OpenGL version.

**16. OpenGL version with SGI V6 Graphics (or higher)**

The graphics do not always get properly updated when running the OpenGL version on a SGI with V6 graphics (or higher), and occasional crashes are found to occur. Starting Mentat with the command line option `-glflush` may solve this problem. The only other known workaround is to use the X version.

**17. Zooming in on Analytical Bodies in Contact may show separation**

When zooming in on an analytical contact body, it may appear as though it is not in contact even though CONTACT STATUS shows that it is in contact. This is because the drawing of the analytical contact is approximate, no matter how high the plot settings are set for CURVES or SURFACES.

**18. Linux client and SGI graphical server**

The MSC.Marc Mentat window will not appear if MSC.Marc Mentat is being run over a network on a Linux machine using a SGI machine as graphical server. This can be solved by starting MSC.Marc Mentat with the command line option `-xfdb false`.

**19. The damping properties are not written to the input file for composite elements**

For composite elements, damping properties are not written to the MSC.Marc input file. They must be entered manually, using either EDIT INPUT FILE or ADDITIONAL INPUT FILE TEXT (model definition section).

# V. Troubleshooting Tips

## MSC.Marc 2005

1. **Contact**

   i. 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.

   ii. If it is an old input using “end of increment penetration check” or “increment splitting” option in contact (i.e. 7th field in the 2nd block of the CONTACT option is 0), then it should be changed to iterative penetration check (i.e. 7th field in the 2nd block of the CONTACT option should be 3). This is the default for the MSC.Marc Mentat 2005 version. The increment splitting option is not supported anymore. Any old input file read into MSC.Marc Mentat, or existing model file when written out in MSC.Marc 2005 format will automatically have this value changed.

   iii. 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).

   iv. 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 MSC.Marc Mentat 2005.

   v. 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 new bilinear friction model in MSC.Marc 2005 generally gives better performance.

   vi. 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.

   vii. 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.

viii. 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.

ix. 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 GUI / input deck should correspond to DOF 1 (in 2-D) and DOF 1, 2, and 3 (in 3-D).

x. 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.

xi. A useful aid for trouble-shooting contact problems is use PRINT,5 parameter in the input deck (in MSC.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.

2. **Load Stepping**

i. 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.

ii. 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.

iii. 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.

iv. 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.

v. 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 in 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.

3. **Materials**

i. 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).

ii. 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 *MSC.Marc Volume A: Theory and User Information*).

iii. 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:

a. flag a nonpositive-definite solution. This usually allows the solution to proceed without impacting the super linear convergence characteristics of the scheme

b. change the flag for the tangent scheme to 3 instead of 1 on the CREEP parameter card. This undocumented flag deactivates the steady-state creep predictor in cycle 0. While this avoids the non-positive definite system, it could impact the convergence characteristics of the solution.

4. **Remeshing**

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

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

      b. 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.

      c. very thin section and large penetration – this can also cause mesher failure as projection of new nodes to the contact surfaces becomes difficult.

      d. 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.

   ii. If there are questionable results:

      a. 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.

      b. due to data mapping, the results in the remeshing increment may show some discontinuity. This is normal.

   iii. Selection of appropriate meshers:

      a. 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.

      b. 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.

5. **Restart**

   i. 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:

      a. switch to table driven input procedure

      b. shift the X-axis of tables in the GUI and write out the portion that remains to be analyzed

      c. 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.

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

**6. Memory Issues in Large Problems**

**i.** To run large problems that address over 2 GB of RAM, you will need the 64-bit version of MSC.Marc running on a 64-bit operating system. To run problems that require more than 8 GB of RAM, consider the following.

There is a 8 GB limit on memory in MSC.Marc. However, this is a limit on some operations within the program and not the total model size. Assuming you have more than 8 GB of physical RAM available, you can run jobs that require more than 8 GB of RAM by trying the following in the order given:

a. If it is a parallel analysis, then decompose the model into domains such that each domain is smaller than 8 GB.

b. Use ELSTO in the parameters section. This writes element quantities to disk. You will need ample free disk space.

c. If there are no Lagrange multipliers (gap or Herrmann elements) or use of structural elements (beams or shells), then switching from a direct sparse solver to an iterative sparse solver would help tremendously both in terms of memory and speed.

d. Use solver 2, preconditioner 3 (i.e. the diagonal preconditioner). Since this solver uses less memory, the amount of memory required may decrease to less than 8 Gbyte. However, this will decrease the computational efficiency as well.

e. Try solver 8 with optimization type 11, that is, the Multifrontal Sparse solver with METIS bandwidth optimization 11 and is now a default in MSC.Marc Mentat.

f. On 64-bit SGI machines, try solver 6, the hardware sparse solver.

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.

Finally, make sure MSC.Marc is configured to allow a full 8 GB of memory allocation. This can be verified by looking at the value of MAXSIZE at the end of the *include* file. The *include* file is in the tools directory. MAXSIZE should be set equal to 2000.

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

**iii.** The SIZING parameter specifies how much memory MSC.Marc should allocate *initially*. If the sizing card specified is too small, MSC.Marc will automatically reallocate memory. For most problems, you will not have to adjust the option, simply let MSC.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 MSC.Marc fill it up as your job progresses.

iv. On the Windows operating system for jobs using the multi-frontal solver (solver8), it may be advantageous to allocate memory for the solver workspace initially. This can be performed using the new parameter PREALLOC. This may be done using the JOB→...→JOB PARAMETERS→SOLVER menu. The advantage of this is while memory may be wasted, there will be sufficient memory for the decomposition phase. In such problems, it may also be useful to activate the out-of-core element storage option (ELSTO).

**7. User subroutines**

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

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

ii. 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.

iii. 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.

**8. OpenGL**

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

i. mentat -*glflush*

ii. mentat -*ss* off

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

# VI. Web Updates for Bug Fixes

As a part of continued commitment to provide the best service to the users of our products, the web based system will be utilized to provide rapid availability of the updates to the released products.

The revisions containing minor enhancements and bug fixes to the MSC.Marc 2005 release (as well as bug fix list) will be available at:

`http://www.mscsoftware.com/support/software_updates/index.cfm#marc`

The announcement for web update will be made on the MSC.Marc home page as well as MSC support page. Appropriate links to this site in the MSC.Marc home page and MSC support page are also available for navigational convenience. The updates to existing MSC.Marc product will be available on ‘as needed’ basis (**please check the MSC.Marc homepage or URL above for new updates**).

Please click on the *readme.txt* before proceeding with the download. The downloaded archives will require passwords to decrypt them. These passwords can be obtained from your local support personnel.

# VII. List of Build and Supported Platforms

## MSC.Marc 2005

| Vendor | OS | Hardware | FORTRAN Version | C Version | Parallel Enabled | Default MPI | Also Works On |
|---|---|---|---|---|---|---|---|
| HP-Compaq (DEC) | Tru64 5.1 | Alpha | f90 5.5 | cc 6.4 | yes | MPICH[1] | |
| HP (64-bit)[2] | HPUX 11.0 | PA2.0 | f90 2.4 | A.11.01.20 | yes | HP MPI 1.06 | |
| HP (64-bit)[2] | HPUX 11.22 | Itanium 2 | f90 2.6.2 | A.05.41 | yes | HP MPI 1.08 | |
| IBM (32-bit) | AIX 4.3.2 | RS/6000 | XLF 7.1 | cc 3.6.6 | yes | MPICH | |
| IBM (32-bit) | AIX 4.3.2 | RS/6000 SP | XLF 7.1 | cc 3.6.6 | yes | IBM POE 3.1 | |
| IBM (64-bit) | AIX 5.2 | RS/6000 | XLF 8.1.1 | cc 6.0.0 | yes | MPICH[1] | IBM POE 4.1 |
| SGI (mips4 32-bit)[2, 3] | IRIX 6.5 | R12000 | f77 7.2.1 | cc 7.2.1 | yes | MPICH[1] | |
| SGI (mips4 64-bit)[2, 3] | IRIX 6.5 | R12000 | f77 7.2.1 | cc 7.2.1 | yes | MPICH[1] | |
| Sun (32-bit) | Solaris 2.8 | UltraSparc II | f90 7.0 | cc 5.4 | yes | MPICH[1] | |
| Sun (64-bit) | Solaris 2.8 | UltraSparc III | f90 7.0 | cc 5.4 | yes | MPICH[1] | |
| RedHat 7 | Linux (32-bit) | Intel Pentium or equiv. | pgf77 4.1-2 | gcc 2.96 | yes | MPICH | |
| RedHat 9 | Linux (32-bit) | Intel Pentium or equiv. | pgf90 5.0-2 | gcc 3.2.2 | yes | MPICH | ScaMPI |
| RedHat AW 2.1 | Linux (64 bit) | Itanium 2 | Intel 7.0 | Intel 7.0 | yes | MPICH[1] | ScaMPI |
| Intel | Windows 2000 | Intel Pentium or equiv. | Compaq Visual Fortran 6.6b | Microsoft Visual C++ 6.0 | yes | MP-MPICH | Windows XP |
| Intel | Windows 2000 | Intel Pentium or equiv. | Intel Fortran 8.0 | Microsoft Visual .NET 2003 C++ | yes | MP-MPICH | Windows XP |

[1] Hardware MPI version also available (via *maintain* in */tools* directory).
[2] Support Solver 6.
[3] Support multi-threading.

**Note:** For most platforms, the default MPI setting for this release uses MPICH, with the option of switching to other specific MPI when so desired. The exceptions are for the following platforms:

*IBM AIX 4.3.2 RS/6000 and RedHat 7, where only MPICH MPI is available.*

*IBM AIX 4.3.2 RS/6000 SP, and HPUX 11.0 (64-bit) and HPUX 11.22 (Itanium 2), where only hardware MPI is available.*

## MSC.Marc Mentat 2005

The following platforms and operating systems are supported by MSC.Marc Mentat 2005.

| Vendor | OS | Hardware | Also Works On | OpenGL[1] |
|---|---|---|---|---|
| HP/Compaq/DEC | Tru64 5.1A | Alpha | | y |
| HP (64-bit) | HPUX 11.0 (64 bit) | PA2.0 | | y |
| HP (64-bit) | HPUX 11.22 (64 bit) | Intel Itanium 2 | | y |
| IBM (32-bit) | AIX 4.3.2 (32 bit) | RS/6000 | AIX 4.3.3 or later | y |
| IBM (64-bit) | AIX 5.2 (64 bit) | RS/6000 | | y |
| SGI (mips3 32-bit) | IRIX64 (32 bit) | Mips3 | IRIX | y |
| SGI (mips4 64-bit) | IRIX64 6.5 (64 bit) | Mips4 | | y |
| Sun (32-bit) | Solaris 2.8 (32 bit) | V7 | Solaris 2.9 | y |
| Sun (64-bit) | Solaris 2.8 (64 bit) | V9 | Solaris 2.9 | y |
| Intel | Windows 2000sp1 | Intel Pentium | Windows XP | y |
| RedHat 7.1[2] | Linux 2.4.2 (32-bit) | Intel Pentium | | y |
| RedHat 9.0 | Linux 2.4.20 (32-bit) | Intel Pentium | | y |
| RedHat AW2.1[2] | Linux 2.4.18 (64-bit) | Intel Itanium 2 | | y |

[1] See OpenGL Compatibility table on the following page.
[2] The Solids version of MSC.Marc Mentat is not supported in this build.

## OpenGL Compatibility

When running over a network, the following combinations of client machine (where MSC.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 visa versa).
[5] Some buffering problems may occur when changing workspaces.

# VIII. List of Dropped Platforms

## Dropped Platforms

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

- HP-UX 10.20
- HP-UX 11 (32-bit)
- SGI mips3
- Compaq Visual Fortran 6.0A (only Compaq Visual Fortran 6.6B or higher will be supported)
- DEC OSF1 4.0 (higher OS is supported)
- Sun Solaris 2.6 (higher OS is supported)
- Windows NT (only Windows 2000 and Windows XP will be supported)

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

- IBM AIX 4.3.2 (higher OS will be supported)
- RedHat 7 & 9 (Enterprise version 3.x and higher will be supported)
- IBM, SGI, and Sun 32 bit
- Compaq Visual Fortran 6.6B

All FORTRAN-77 versions will be dropped in the next release and will be replaced with FORTRAN-90 versions.

# IX. Important Notes

## MSC.Marc 2005

**1.** The startup script for MSC.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 MSC.Marc Mentat.

**2.** When running any of the examples in the *MSC.Marc User's Guide* or *MSC.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:

(1) Using the command line option `-nthreads`.

Example:

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

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

(2) 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 MSC.Marc Mentat, the variable needs to be set before MSC.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 MSC.Marc is delivered with MPICH (public domain 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 *MSC.Marc and MSC.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 is the 64-bit HP version which fully supports the network parallel analysis with HP-MPI.

   **Note:** The Linux RedHat9 and RedHat AW 2.1 versions support ScaMPI.

5. Installation related:

   i. If you get an error message of `f77 not found` or `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.

   ii. 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:

      a. Try relinking the version first by executing the `make_marc` script in the `marc2005/tools` directory and run the job with and without user subroutines.

      b. If the problem persists, check if the `.so` file exists in the `marc2005/lib/lib_shared` directory. If it does exist, uncomment the following two lines in the `run_marc` script under `marc2005/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.

      c. If the `.so` files do not exist in the `marc2005/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 MSC.Marc job, the output will automatically be written to the directories specified in the hostfile. For instance, when running a 4 domain MSC.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, MSC.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.

# X. Platform Specific Notes

## Various Machine Notes

**1. SGI Machines:**

(1) 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 MSC.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.

(2) The current version of MSC.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.

(3) 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 at least as large as that specified with the MAXSIZE value in the `include` (or `include.bat`) file. 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:**

(1) 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.

(2) 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.

(3) 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-SP Machines:**

MP_EUIDEVICE=css0

This specifies that the tasks will use the High Performance Switch for communication.

**Note:** A group of workstations connected by an ethernet network would use:

MP_EUIDEVICE=en?

where:

? specifies the network device adapter [ 0, 1, 2, ..., n ]

MP_EUILIB=ip

This specifies that the task will use "ip" protocol when sending messages to each other. Not the best performance, but always gets through the switch.

MP_EUILIB=us

This gives the best performance over the switch, but causes delays in starting jobs on older versions of PSSP.

**Note:** A group of workstations communicating over a network would have to specify "ip" since "us" is only available for communication over the switch.

#MP_RESD=yes

No longer used for job scheduling with LoadLeveler. Required for older version of PSSP which uses the Partition Manger to schedule jobs.

MP_RMPOOL=1

This specifies a “Pool” of nodes that LoadLeveler will use for scheduling the tasks if the file *host.list* does not exist in the directory where the job starts.

MP_HOSTFILE=”NULL”

This specifies the file that contains the list of nodes to schedule the tasks on, will have the default name *host.list* if it exist.

#MP_HOSTFILE=$DIR/tools/host.list

Allows the user to specify any file name and its location, instead of *host.list* in the directory where the job starts.

MP_INFOLEVEL=0

Specifies the amount of information that LoadLeveler/POE gives the user about his job. The zero specifies “none” and the user would not know what happened if his job failed. Use a value of 2 (default).

4. **IBM Machines:**

   The OpenGL version of MSC.Marc Mentat may not operate properly on some IBM platforms. The runtime libraries patch 4.3.3.25 is required. The type of graphics adapter may be displayed using the `lsdisp` command.

5. **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.

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

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

7. **Linux RedHat 9 and RedHat AW 2.1:**

   In order to use ScaMPI for parallel jobs, it must be obtained from
   Scali (`http://www.Scali.com`).

   The MSC.Marc version can be switched using tools/maintain just as for hardware MPI on other UNIX platforms. The ScaMPI version supports single multiprocessor machines as well as the following network types:

   – Ethernet

   – Fast Ethernet

   – Gigabit Ethernet

   – Myrinet

   – SCI

   – Hyperfabric

# XI. Security

## Security Notes

The 2005 release stores the license manager (lmgrd) and the vendor daemon in a directory named MSC.Licensing\9.2 for Windows 2000/XP 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.

| | | |
|---|---|---|
| i. | MARC | license required to run one single processor job or one instance of a multiple processor (parallel) job. |
| ii. | 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). |
| iii. | MARC_Mesh2D | license required for each run requiring automatic 2-D remeshing feature in MSC.Marc. |
| iv. | MARC_Mesh3D | license required for each run requiring automatic 3-D remeshing feature in MSC.Marc. |
| v. | MARC_ShapeMemory | license required for each run using shape memory model. |
| vi. | MARC_MetalCutting | license required for each run modeling metal cutting operation. |
| vii. | MARC_CrossSection | license required for each run using the Cross Section option. |
| viii. | MARC_Electrical | license required for Joule-Mechanical, Coupled Electrostatic-Structural, and Piezoelectricity. |
| ix. | MARC_Viewfactor | license required for calculation of radiation viewfactors using the viewfactor program. |
| x. | Mentat | license required for each instance of MSC.Marc Mentat. |
| xi. | MARC_Hexmesh | license required for each instance of Hexahedral mesher. |
| xii. | Mentat_ACIS | license required for each instance of ACIS when working (import/export) with ACIS based models. |
| xiii. | Mentat_ITI_Access | license required for each instance of, or exporting a file using the DXF, IGES, or VDAFS translators. |
| xiv. | Mentat_CMOLD | license required for each instance of CMOLD when working (import/export) with CMOLD based models. |

The following types of licenses are available with the products:

i. Nodelocked, counted
ii. Nodelocked, uncounted
iii. Floating (concurrent)
iv. Campus
v. Demo

Please refer to the *MSC.Marc and MSC.Marc Mentat Installation and Operations Guide* for more information on security or contact your local MSC representative.