MSC.Patran Marc Preference™

The power of MSC.Marc™, the strength of MSC.Patran™

OVERVIEW

MSC.Patran is a world-class finite element modeling pre- and post-processor, which is tightly integrated with the MSC.Marc FEA™ program, allowing data to be defined interactively through a powerful graphical user interface. All aspects of creating a MSC.Marc model from material, property, and load definitions to analysis submittal and results postprocessing are completely controlled through MSC.Patran 2003.

MATERIAL MODELS

Most material constitutive models are supported through MSC.Patran's materials application including isotropic, orthotropic and anisotropic materials. Individual property values may vary with strain, strain rate, time, frequency, or temperature. MSC.Patran also provides experimental data fitting with graphical feedback for hyper- and visco-elastic material models which includes the capabilities to import the test data, perform calculations to determine best-fit coefficients for the selected constitutive model, automatically plot the comparison between the test data and calculated coefficients, and then create the material model.

Visualization of a head gasket and part of the engine block.

Elastic (3-D and 2-D conditions)
- Plane stress / thin shell
- Plane strain / axisymmetric
- Thick shell
- Axisymmetric with twist
- Axisymmetric shell

Plastic
- Isotropic
- Elastic-plastic
- Perfectly plastic

√ *Enhancement in 2003*

PRODUCT LINE

MSC.Patran™

CAPABILITIES

- Full support of nonlinear structural, thermal, and coupled analysis
- Visualization of all output quantities
- Animation of transient and nonlinear behavior
- CAD Integration based on the power of MSC.Patran
- Mesh generation of complex models
- Adaptive meshing

BENEFITS

- Interoperability with MSC.Nastran™ and other analysis codes
- Customization using PCL
- World class pre- and postprocessing
- Easy product component simulation
- Better understanding of product behavior

MSC SOFTWARE®
SIMULATING REALITY

Rigid-plastic
Hardening rules
- Rate power law
- Kinematic
- Johnson-Cook
- Combined
- Kumar
- Power law

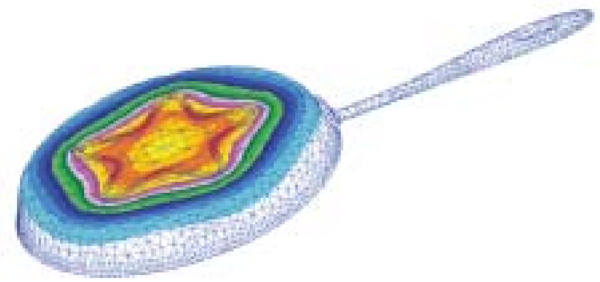
Yield criteria
- von Mises
- Linear Mohr-Coulomb
- 2-1/4 Cr-Mo ORNL
- Reversed plasticity ORNL
- Parabolic Mohr-Coulomb
- Full Alpha reset ORNL
- Buyukozturk concrete
- Generalized plasticity
- Oak Ridge National Lab (ORNL)

Strain rate methods
- Piecewise linear
- Cowper-Symonds

Hyperelastic
Neo-Hookean
Mooney-Rivlin
Jamus-Green-Simpson
Ogden
Foam
Arruda-Boyce
Gent
√ Experimental data fitting

Viscoelastic
Creep
Damping
Failure criteria
Hill
Hoffman
Tsai-Wu
Maximum strain
Maximum stress

Thermal
√ Composites - 2-D and 3-D laminated and gaskets



Thermal contact between stove burner and a pan.

ELEMENT LIBRARY

Most MSC.Marc structural and thermal elements are defined through MSC.Patran's element property application and special elements such as rigid tying elements are defined through the elements application. Properties can be assigned to elements directly or to associated geometry as desired.

0-D elements
Mass Elements
√ Linear and nonlinear grounded spring/damper

1-D elements
Elastic beams (types 52, 98, 31)
Thin walled beam (types 14, 25, 26, 78)
√ Linear and nonlinear spring
Damper
Gap (types 12, 97)
Cable (type 51) planar
Axisymmetric shell (types 1, 87, 88, 89, 90, 15)
2-D Beam (types 5, 45, 16)
Link (thermal) (types 36, 65)
Truss (types 9, 64)
√ Plane strain membrane rebar (types 165, 168)
√ Axisymmetric membrane rebar (types 166, 167, 169, 170)
√ Full support of the MSC.Patran beam library of standard and arbitrary shapes.

√ *Enhancement in 2003*

2-D elements

Thick shell (types 22, 75, 140)
Axisymmetric
- Standard formulation (types 2, 10, 28, 126 38, 40, 42, 132)
- Herrmann (types 82,156, 33, 129)
- Herrmann/Red. Integration (types 59, 119)
- Herrmann /Twist (types 66, 83)
- Reduced integration (types 55, 116, 70, 122)
- Twist (type 20, 67)
- Laminated composite (types 152, 154)
- Fourier (type 62)
- Herrmann/Fourier (type 63)
- Reduced integration/Fourier (type 73)
- Herrmann/reduced integration/Fourier (type 74)
- Bending (types 95, 96)
- Semi-infinite (types 92, 94, 102, 104)

Plane strain
- Standard formulation (types 6, 11, 27, 125, 37, 39, 41, 131)
- Reduced integration (types 54, 115, 69, 121)
- Herrmann (types 32, 80, 128, 155)
- Herrmann/Red. Integration (types 58, 118)
- Generalized (types 19, 29)
- Generalized reduced integration (type 56)
- Generalized/Herrmann (types 34, 81)
- Generalized/Herrmann/Red. Integration (type 60)
- Laminated composite (types 151, 153)
- Semi-infinite (types 91 93, 101, 103)

Plane stress
- Standard formulation (types 3,26, 124)
- Reduced integration (types 53, 114)

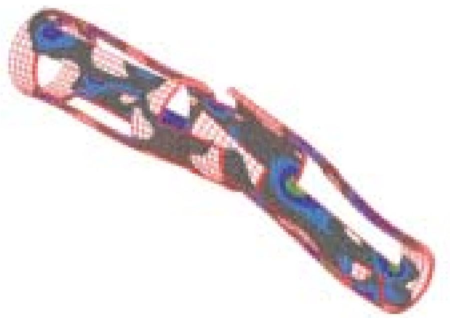
Membrane (types 18, 30)
Shear panel (type 68)
√ Membrane rebar (types 147, 148)

3-D elements

Standard formulation (types 7, 21, 127, 134, 43, 44, 133, 135)
Reduced integration (types 57, 117, 127, 134, 71, 123, 135)
Herrmann (types 35, 84, 130, 157)
Herrmann/Red. Integ. (types 61, 120, 130)
Laminated composite (types 149, 150)
Semi-Infinite (types 107, 108, 105, 106)

Rigid tying elements

Explicit - SERVO LINK
Tying types 31, 32, 33, 34, 100, 1-6, 102-506, 26, 49, 50, 52, 53, 80



This ensemble represents two telescoping, cutout cylindrical pieces, the outer one fixed to a wall; the inner one fixed to a rigid surface that acts as a driver. The driver rotates about a horizontal axis at the center of the cylinder on the wall, carrying the glued section of the inner piece.

LOADS AND BOUNDARY CONDITIONS

All loads are defined graphically and can be static or time dependent. They can be associated to finite elements or geometry as desired. Multiple load cases can be created and loads associated accordingly. This functionality is found under the Loads and BC's application and the Load Cases application.

Displacements	Initial Velocity
Forces / Moments	Distributed Load
Pressure	Convection
Nodal Temperature	Heat Flux
Elemental Temperature	Heat Source
Inertial Load	Initial Temperature
Initial Displacement	√ Thermal Radiation
√ Initial Stress/Strain	√ Convective Velocity

CONTACT DEFINITIONS

MSC.Patran treats contact definitions as a boundary condition and thus are defined under the Loads and BC's application. Each contact body is defined separately. No special elements are required or any contact pair definitions. A contact table controls the definition of which bodies are allowed to contact which bodies. By default all bodies may come in contact with all other bodies including self-contact.

√ *Enhancement in 2003*

The contact table also provides explicit control of the contact interaction parameters (such as distance tolerance and friction coefficient) between individual body pairs.

Friction control parameters

Separation and penetration controls

Contact Table Support

- Touching, glued or null contact between pairs
- Heat Transfer Coefficient Between Bodies
- √ Control of Contact Body Numbering
- Separation Force
- √ Body release and sudden/gradual force removal
- √ Stress-Free Initial Adjustment
- Distance Tolerance
- √ Delayed Slide-Off
- Interference Closure
- Friction Coefficient
- Retain Gaps/Overlaps

Deformable bodies

- √ Smooth Analytic Surface representation or faceted deformable bodies
- Exclude portions of bodies that do not come into contact

Rigid bodies

- Visual aids to determine contact side and inward directions
- Velocity and position controlled rigid bodies
- Force and moment controlled rigid bodies
- √ Preview of rigid body motion
- √ Visualization of rigid body motion during post processing
- Discrete or geometric (NURBS) rigid body definition

GENERAL ANALYSIS SETUP

MSC.Patran directly submits the MSC.Marc input deck for analysis. The following general capabilities are supported for most analysis solutions available.

Local and remote submittals

Job setup and monitoring control

√ Edit input deck

√ Real-time job monitoring

√ Analysis manager support

View status, output, and log files

Keyword search of output file

√ Job submittal to LSF

Keyword search of output file

Solver options

Direct profile	Direct sparse (default)
Iterative sparse	Hardware sparse
Multi-frontal sparse	

√ Support for Domain Decomposition with manual, semi-automatic or automatic (using Metis) domain creation

Direct text input - allows any keyword support in:

Parameter section

Model definition section

History section

Convert any MSC.Patran group to:

Element set

Node set

Restart parameters

User subroutine file or compiled program selection

Adaptive meshing setup

Local refinement

Global remeshing

Visualize adapted mesh results

3-D Global remeshing

Load step capability

Multiple analysis types in single run

Change convergence criteria or load stepping procedure

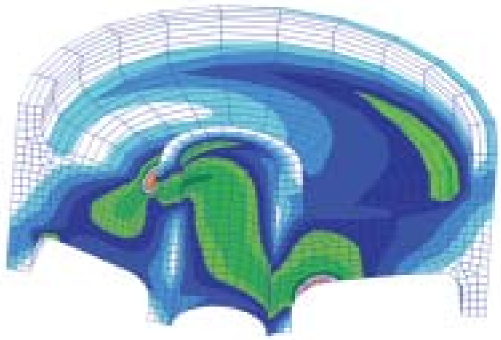
Element activation/deactivation

Temperature loading via previous thermal analysis file specification

√ Axi-symmetric to 3-D model results mapping

√ Cyclic symmetry

Nodal/elemental output requests (POST)



Rubber bushing subjected to a loaded shaft.

√ *Enhancement in 2003*

ANALYSIS SOLUTIONS

The MSC.Marc Preference™ in MSC.Patran supports all structural, thermal and coupled thermal- mechanical solutions. Multiple analyses can be set up in a single database and each analysis can be associated to multiple analysis Load Steps to perform the most complicated of analyses.

Statics
- Fixed load stepping (AUTO LOAD)
- √ Thermal adaptive stepping
- Adaptive load stepping (AUTO STEP)
- Arc length methods (AUTO INCREMENT)
- √ Super-Plastic Forming (pressure rate controls strain rate)
- Automatic load step cutback feature
- √ Iteration parameter control
- Automatic convergence control
- Eigenvalue pertubations

Normal modes and buckling
- Inverse power sweep
- Lanczos

Transient dynamics
- Fixed time stepping (DYNAMIC CHANGE)
- √ Adaptive time stepping (AUTO STEP)
- Linear and nonlinear solutions
- Direct and modal formulations
- Large/small strain/displacement
- Linear and nonlinear solutions

Frequency response

Spectrum response

Creep
- Explicit and implicit procedures
- Multiple creep methods
- Adaptive time stepping (AUTO CREEP/STEP)
- Fixed time stepping (CREEP INCREMENT)
- √ √ Thermal Adaptive time stepping (AUTO THERM CREEP)

Body approach for multi-step forming

Steady state heat

Transient heat
- Fixed time stepping (TRANSIENT NON AUTO)
- √ Adaptive time stepping (AUTO STEP)
- Thermal adaptive time stepping (TRANSIENT)

√ Coupled thermal-structural
- Psuedo-static/transient heat
- Creep
- Fixed time stepping (TRANSIENT NON AUTO)
- Thermal adaptive time stepping (TRANSIENT)
- Adaptive time stepping (AUTO STEP)

POST-PROCESSING

MSC.Patran is a world class post-processor for interpreting finite element results. There are two ways of accessing MSC.Marc results by either importing them directly into the MSC.Patran database or simply attaching the results file through direct results access (DRA).

Attach results file (DRA)

Supports post code revisions 2000 and 2001

Model import (nodes, element, coordinates)

NURB geometry import (rigid bodies)

Creates groups by property IDs

Results remain in result file

√ Adapted meshes visualization

Import results file
- Discrete rigid body animation
- Model import (nodes, element, coordinates)
- Results filter on increments, element, nodes and result types

Binary compatibility

Reads t16 and t19 post files

√ Quick plot ability to sequentially step through multiple load cases

This is a coupled structural-thermal analysis of a friction clutch. Rigid surfaces first compress the clutch and then rotate relative to each other.

√ *Enhancement in 2003*

Post codes (structural/thermal) (most post codes > 300 supported plus those shown below):

Scalar elemental post codes

- Equivalent strains(7, 8, 27, 28, 37, 127)
- Strain through the thickness (49)
- Temperatures (9, 10)
- Equivalent stresses (17, 47)
- Energy densities (48, 58, 68)
- State variables (29, 39)
- Hydrostatic stress (18)
- Failure indexes (91-97)
- Interlaminar stresses (108, 109)
- Thickness/Volume (20, 78)

Vector elemental post codes

- Temperature gradient (181-183)
- Flux (184-186)

Tensor elemental post codes (>300)

- Cracking strain (381)
- Thermal strain (371)
- Creep strain (331
- Total strain (301)
- Elastic strain (401)
- Stresses (311,341)
- Plastic strain (321)

Nodal post codes (1-40)

- Displacements
- Temperature
- Velocities
- Flux
- Accelerations
- Contact normal stress/force
- Forces/moments
- Contact status/touched body
- Modal mass
- Herrmann variable

Global variables

- Increment
- Mode
- Time
- Critical load factor
- Frequency
- √ Body variables

INPUT FILE READER

Existing MSC.Marc input files can be read into MSC.Patran. The following can be imported:

Nodes
Most materials
Elements
√ Most element properties
Coordinate frames
√ Rigid/Deformable contact bodies
√ Most loads
Most boundary conditions

To find your local MSC.Software office or to learn more about our company and our products, please contact:

Corporate:

MSC.Software Corporation
2 MacArthur Place
Santa Ana, California 92707 USA
Tel: 1 714 540.8900
Fax: 1 714 784.4056

Customer Care Center:

1 800 642.7437 (U.S. only)
1 978 453.5310 (International)
customer.care@mscsoftware.com

Worldwide Web - www.mscsoftware.com
On-line Purchases - www.engineering-e.com

P3*3-03*Z*MAPR*Z*LT-DAT

MSC SOFTWARE®
SIMULATING REALITY