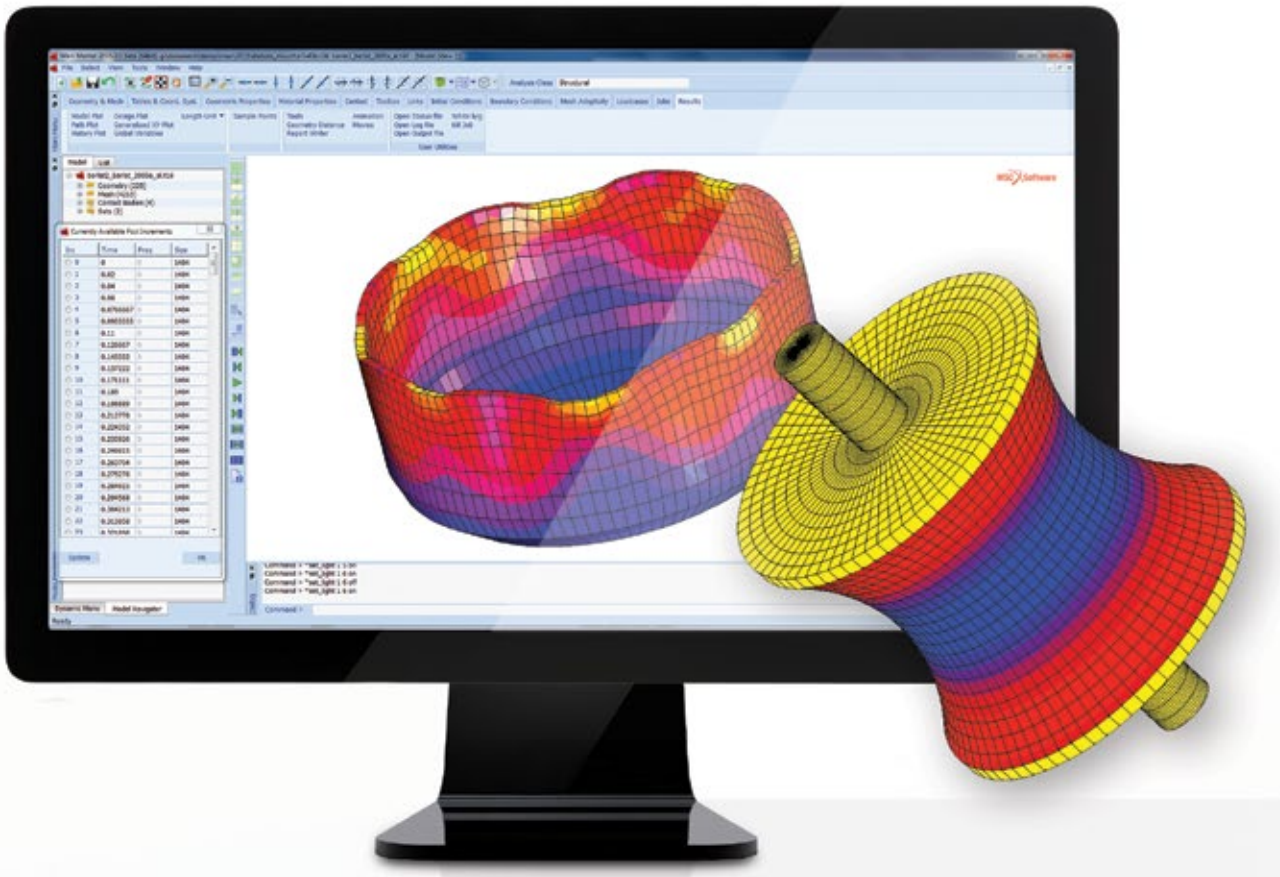
Marc®

Advanced Nonlinear and Multiphysics Analysis



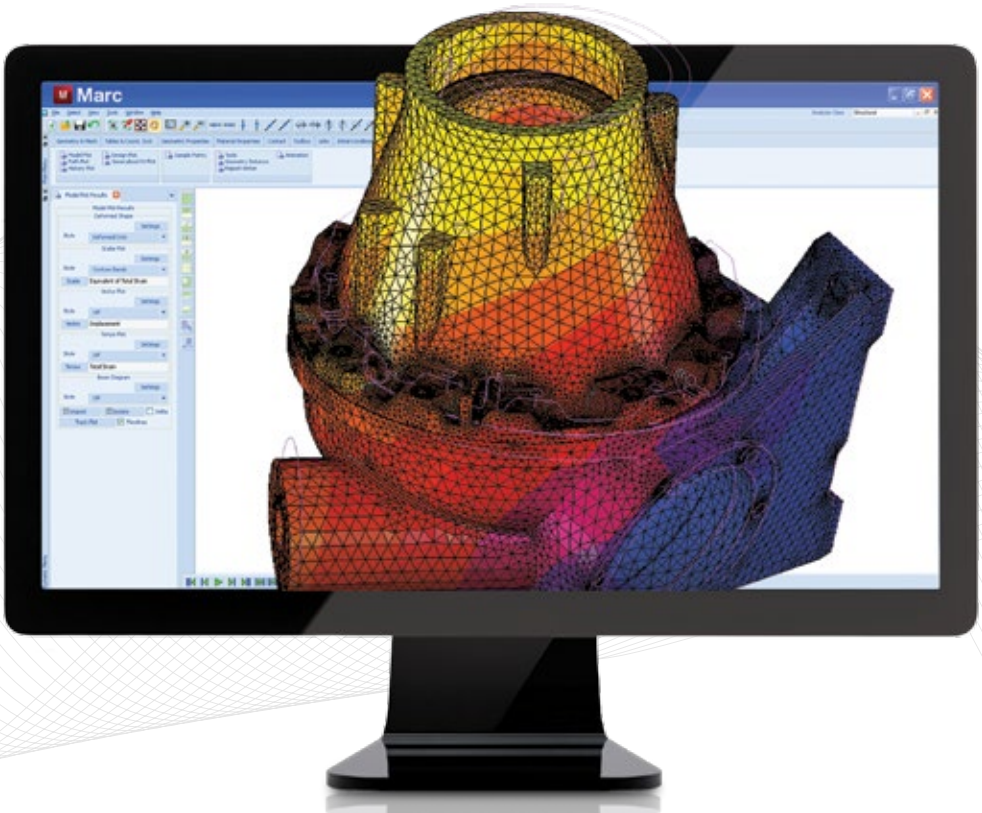
PART OF HEXAGON

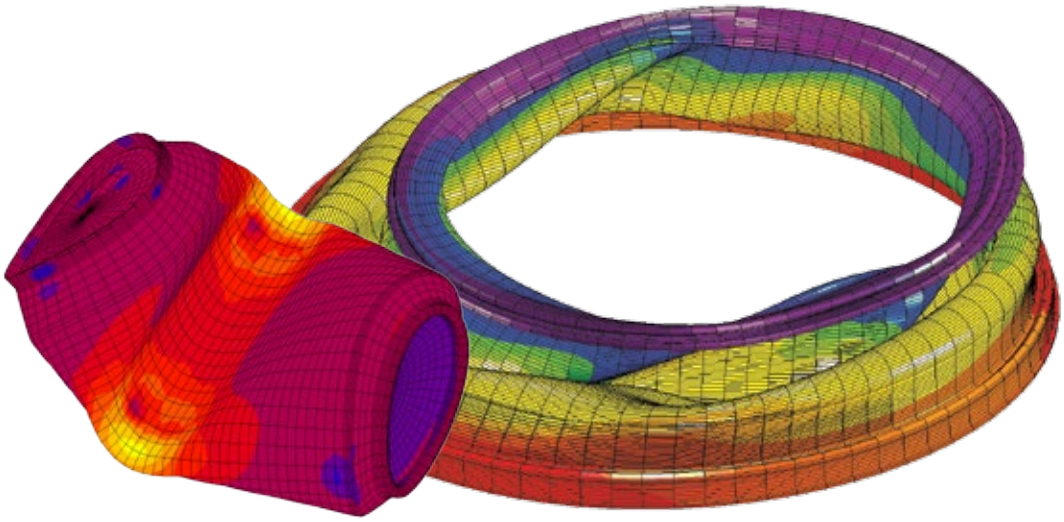
MSC Software®

Nonlinear Analysis for Improved Designs

Nature is nonlinear. Using Marc, accurately capture the inherent nonlinear behavior of your designs to improve product quality, reduce your testing costs, and improve reliability incorporating the true product environment.

Marc, the dedicated nonlinear finite element analysis (FEA) solver from MSC Software, is designed to simulate complex engineering processes incorporating nonlinear behavior of materials, geometry and boundary conditions. Through its innovative simulation tools, Marc offers solutions to your toughest nonlinear problems, saves you time, and improves your productivity.





Why Marc for Nonlinear FEA

Accurate Nonlinear Analysis

Products are designed to withstand multiple loading scenarios under various environmental conditions. Hence, it is important to be able to simulate these conditions in order to get a clearer picture of the product behavior and to improve your designs. Marc offers a broad class of solution procedures that you can use to virtually simulate the desired testing conditions and service loads. The industry proven, state of the art methodologies are designed to provide you with accuracy and efficiency that you require in your development process.

Nonlinear Material Models

The spectrum of engineering materials in use is growing, and so is the need for extensive testing of products designed with these materials. Customizable materials like composites, plastics, elastomers, and new metal like shape memory alloys are helping engineers improve their products while operating under engineering constraints.

Marc provides users the ability to represent a broad range of materials, including metals, shape memory alloys, super-plastic materials, composites, wood, plastics, rubbers, glass, ceramics, concrete, powder metals, and many more. The models incorporate temperature effects, rate effects, phase transformations and damage so they can accurately predict the behavior throughout a product's operating range. Marc also offers an easy approach to implement new material models that are used in state of the art designs and research. So, whether your designs use metals like steel and aluminum or complex materials like composites, solder, or foams, Marc provides you with industry accepted material models to accurately represent their behavior.

Multiphysics and Coupled Analysis for Real World Behavior

As newer materials are used in designs to benefit from their unique properties, it also becomes important to model the physics that control their behavior. For example, piezoelectric materials accumulate electric charge when subjected to mechanical stress, and this phenomenon is used in sensors, actuators, and motors. Also, the phenomenon of Joule heating or resistive heating due to the electric energy converted to heat has multiple applications including cooking plates and automotive defroster grids.

The multiphysics capability of Marc, when combined with its superior nonlinear structural analysis, provides more accurate results leading to better designs of your structural systems. Marc can be used to couple structural, thermal, magnetostatics, electrostatics, induction heating, electromagnetic and fluids (laminar flow with small structural deformation) behaviors. The coupling capabilities are highly useful to improve the accuracy of manufacturing simulations like welding, curing, and forming.

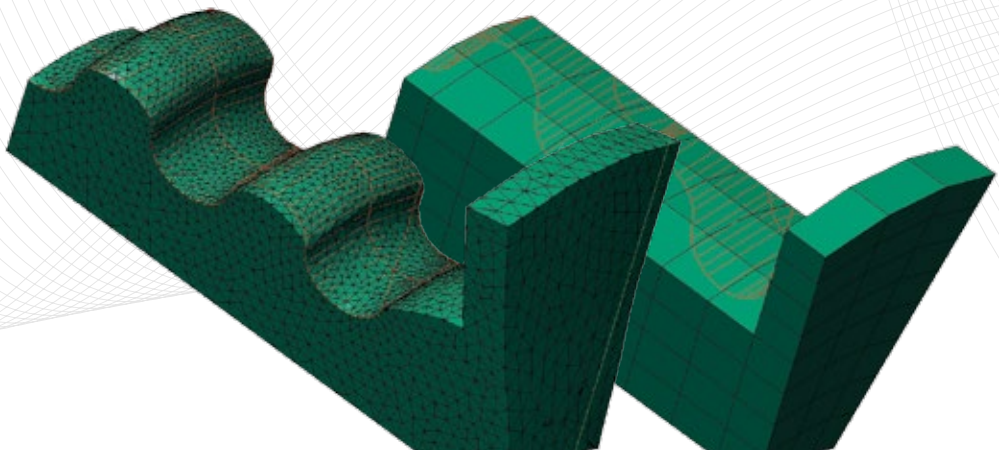
Easy to set up Contact Analysis

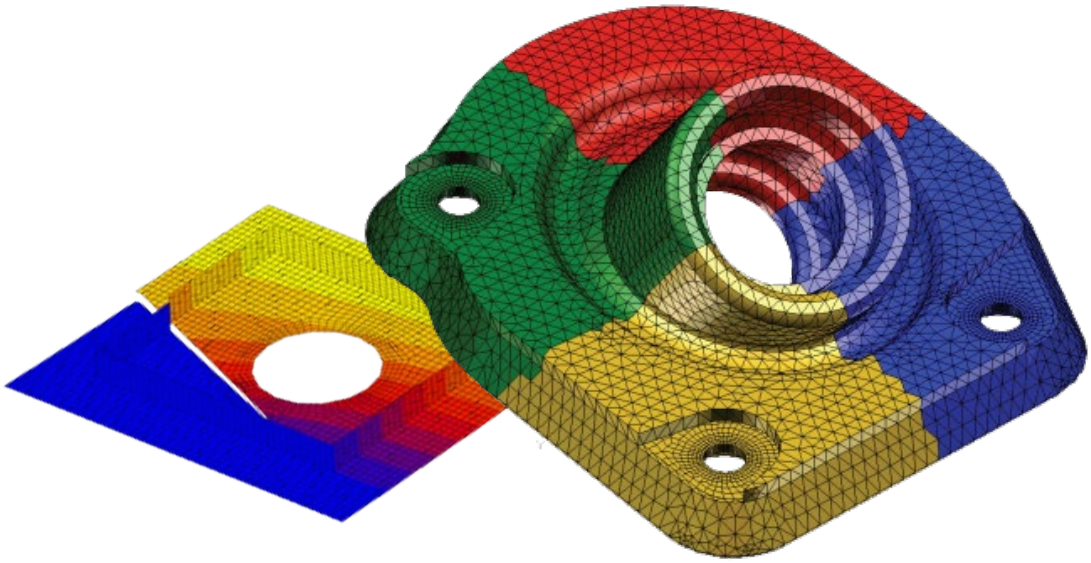
Almost all designs involve interaction with other components within an assembly or interaction between assemblies transferring forces across the contact zones. In a nonlinear analysis, force direction and magnitude along with the contact zone often change continuously. For analysis accuracy, it is critical to study the normal contact stresses and the shear stresses transmitted during the component interaction.

Marc offers a unique approach in tackling this tough problem with the use of smart procedures and easy modeling. Intuitive and easy contact body definitions and automatic contact boundary detection relieve users of the burden to create contact interfaces or define master and slave surfaces. Since Marc is designed to handle both small sliding and large sliding between contact bodies with a single algorithm, users do not need to be concerned about the approach they need to use for their specific problem. Smarter procedures which do not sacrifice performance make for less work for users, while providing improved accuracy.

Failure Analysis for Better Products

Prediction and investigation of failure is an essential step in product design. Knowing what caused failure helps improve future designs and extend product life. Multiple damage models are available for brittle materials such as concrete, ductile metals, and rubber materials. Advanced composite materials provide new challenges. The Failure and Fracture mechanics technology in Marc allows engineers to predict matrix, fiber, and delamination failures. Advanced fracture mechanics capabilities help to predict both crack initiation and crack propagation. These procedures may be used with a wide range of materials and for virtually all geometries. This provides you the tools to ensure reliable, safe designs.





Performance for Improved Productivity

Marc enables higher productivity through solver efficiencies delivered on multiple fronts. First, Marc helps users achieve higher efficiency through state of the art solver technology. With the falling cost of multi-core systems, parallel processing is increasingly accessible to small and medium businesses operating in a desktop environment. To help users make full use of their hardware, Marc provides parallel solvers to everyone at no additional cost. Parallelization can also be used for the element assembly and stress recovery steps to reduce the wall time for the nonlinear analysis. Finally, through its unique implementation of domain decomposition method whereby the model is broken into smaller pieces and solved in entirety on separate processors, Marc goes beyond traditional parallelization techniques common in industry. With this approach, all the steps of FEA process including, input, stiffness matrix assembly, matrix solution, stress recovery, and output are executed in parallel on shared memory or distributed memory systems helping achieve superlinear scalability for your large models. Reducing wall time will result in more efficient simulations and better designs.

Automatic Adaptive Meshing

During manufacturing processes or sealing applications, and fracture mechanics, the material undergoes severe deformation due to the applied loads or contact forces. These deformations can be so large that the finite element mesh could become highly distorted leading to inaccuracy of results. Marc offers an innovative solution to overcome this problem through the use of automatic remeshing.

During the analysis, if elements become severely distorted, Marc automatically creates a new mesh from the deformed boundary. The state of the material (stresses, strains, deformations) along with contact conditions are transferred to the new, well structured mesh, and the analysis is continued. This process is repeated as many times as necessary without user intervention allowing you to solve tougher, complex nonlinear problems.

Customization

Marc provides the ability to customize the analysis software through the use of user subroutines. Over 200 are available to incorporate advanced material models, load and boundary conditions, or element technology. The GUI – can be easily customized using Python.

Easy to Use, Integrated User Interface

Tightly integrated user interface to help you build, analyze, and postprocess the FEA models with ease. Designed with focus on supporting nonlinear analysis, you benefit from time-saving tools and processes implemented in the Marc user interface.

"Benchmark investigations showed clearly that Marc was the best solution in every aspect"

Dr. Martin English,
Hadley Group Technology

Geometry and Meshing

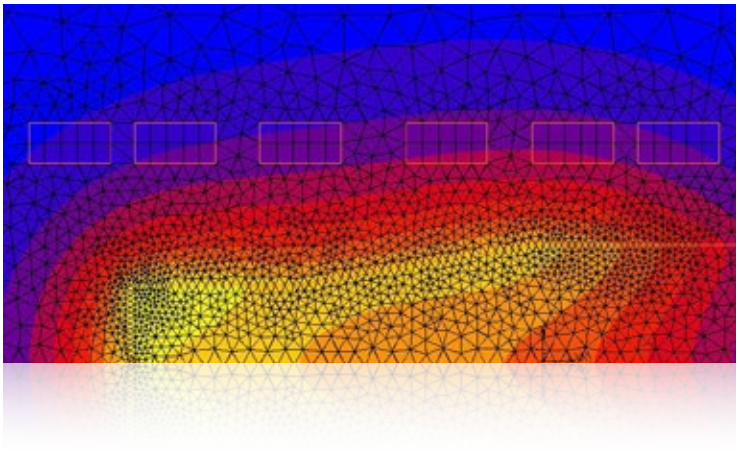
- Import CAD files in neutral formats of ACIS, IGES, Parasolid, STEP, STL & VDAFS
- Import geometry in native CAD formats of CATIA V4, CATIA V5, DXF, Inventor, JT, Pro/ENGINEER, Solidworks & Unigraphics
- Create and edit curves, surfaces, and solids
- Defeature CAD geometry
- Create 1D, 2D, and 3D lower and higher elements using automatic mesh generators
- Refine and edit mesh entities
- Create springs and dashpots for analysis
- Create discrete connecters such as RBE2, RBE3, Servo-links, CBUSH, CFAST, and CWELD
- Insert nodes or elements within other elements
- Check integrity of model

Material Modeling

- Define nonlinear material parameters'
- Curve fit experimental data to obtain parameters for elastomers and metals
- Specify temperature dependent material properties
- Define properties as functions of other independent variables

Loads & Boundary Conditions

- Define required forces, moments, displacements, pressures, and rotations
- Specify boundary conditions relevant to each multiphysics model set up
- Apply boundary conditions to geometric entities to easily transfer to associated mesh
- Define boundary conditions as functions of other independent variables
- Group loads and boundary conditions into load cases easily



Contact

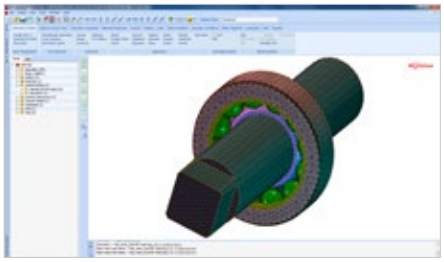
- Easily define deformable and rigid contact bodies for all geometries
- Automatically create contact bodies from CAD assembly models
- Customize contact interaction using contact tables
- Define friction coefficients and other parameters for contact analysis
- Supports large deformation interference fit
- Support contact in multiphysics applications.
- Accurate assessment of transfer loads and fluxes

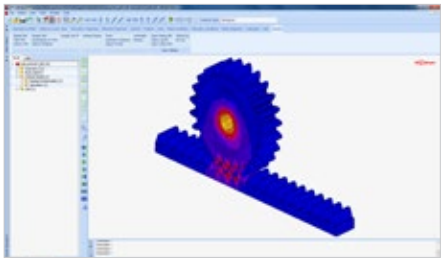
Post-processing

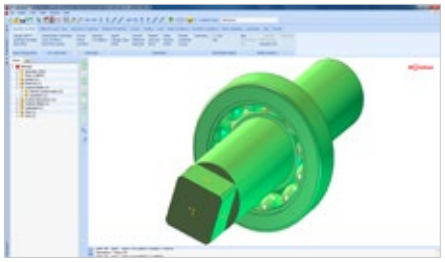
- Obtain plots of results in format you desire including contour plots, contour lines, cutting planes, iso-surfaces, tensor plots, beam plots and more
- View time histories of results save and any arbitrary location
- Use path plot to review the spatial variation of any result along any arbitrary path
- Create images and movies for reports and presentations
- Track material particle flow
- Create reports for results selected by range or Max/Min
- Visualize 3D beam contact
- Visualize of contact stresses

Modeling Tools

- Easily create bolt models and loads for assembly analysis
- Create gas filled closed cavities and analyze their influence on structural response
- Transfer pre-state from one analysis to another
- Map results from axisymmetric analysis to a 3D model
- Create symmetry and cyclic symmetry boundary conditions
- Define welding paths and filler elements for welding analysis
- Activate and Deactivate elements
- Define crack tips for fracture analysis
- Define electrical coil windings for electromagnetic analyses







Nonlinear Structural Analysis

Reduce your physical testing costs with reliable, robust simulation of structural response under a wide spectrum of loading scenarios and environmental conditions.

"Marc gives us confidence in our results and offers a complete list of features. I am very satisfied with the results of our analyses"

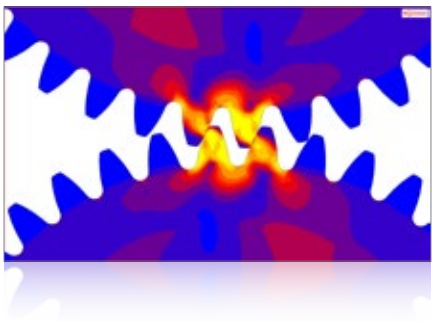
Luigi Piccamiglio, Invernizzi Presse

Static Analysis

- Perform linear and nonlinear static analysis to virtually test your designs
- Include advanced nonlinear material models
- Incorporate both large deformation and large strain behavior
- Accurately model nonlinear boundary conditions including follower force effects, foundations, and contact
- Perform creep simulations to determine the long term response of the structure.
- Perform post-buckling analysis to perform stability studies
- Determine the inertia relief force to balance free structures
- Perform steady-state rolling analysis of tires
- Perform mechanical wear analysis due to friction
- Export or Import DMIG files for compatibility with MSC Nastran
- Perform global-local analysis to better capture local behavior

Dynamic Analysis

- Perform natural mode analysis of structures to determine structural stability under dynamic loads
- Conduct frequency response analysis subjected to harmonic loads or random vibrations to analyze structural performance
- Include advanced damping models that incorporate frequency and deformation dependent damping observed in rubber and plastics
- Obtain insight into dynamic performance of structures through linear and nonlinear transient analysis
- Gain improved accuracy through accurate modeling of contact, nonlinear materials, and loading conditions
- Create Modal Neutral Files (MNF) that may be shared with Adams including nonlinear preload
- Consistent or lumped mass matrix



Multiphysics and Coupled Analysis

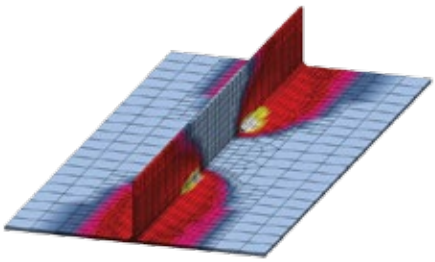
The multiphysics capability of Marc, when used with its superior nonlinear structural analysis, provides more accurate results leading to better and improved designs of your structural systems.

"Marc is a multi-physics tool with capabilities for flow, heat, mechanical stress, and electrical modeling. In addition, the open structure of Marc allows customization - databases, user routines, and so on"

Dr. Moe Khaleel,
Pacific Northwest National Laboratory

Heat Transfer

- Perform steady-state and transient analysis for one-, two-, and three dimensional bodies
- Obtain temperature distributions in a structure for linear and nonlinear heat transfer problems
- Model nonlinearities including temperature-dependent properties, phase transformations, latent heat effect, heat convection in the flow direction, and nonlinear boundary conditions (convection and radiation)
- Fast and accurate calculation of view factors, dependent on deformation
- Simulate thermal degradation of Thermal Protection Systems (TPS) with advanced pyrolysis model
- Perform ablation analysis for space systems, brakes, and bio-medical applications
- Compute heat fluxes across multiple components that come into contact

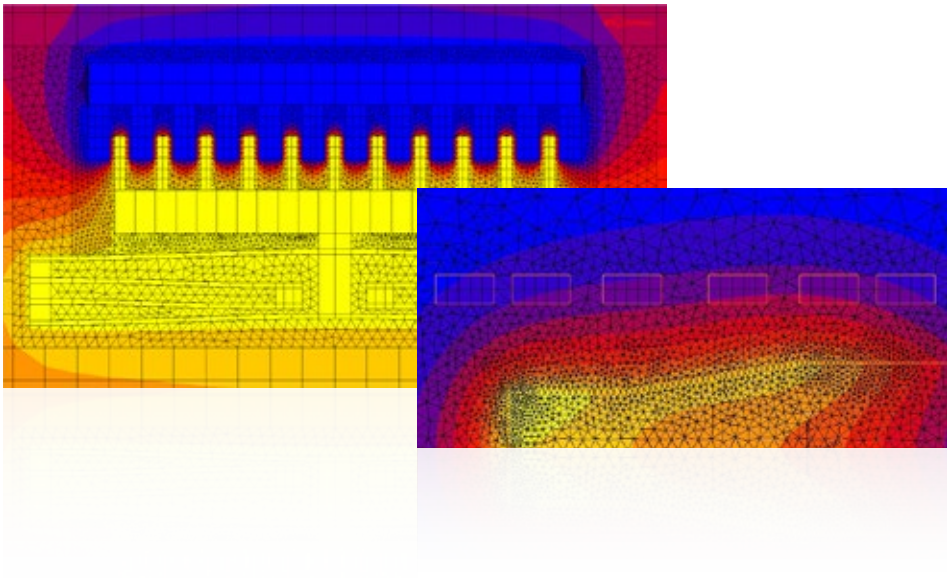


Thermomechanical Coupling

- Simulate manufacturing processes
- Analyze structural response due to temperature changes in the environment and thermal gradients in the structure
- Model heat generation due to plasticity and friction between different components for accurate physics
- Incorporate heat generation due to curing in composite manufacturing.
- Simulate the influence of annealing
- Simulate the effects of changes to thermal boundary conditions due to large deformations

Acoustics and Coupled Acoustic-Structural Analysis

- Simulate window seals
- Perform acoustic analysis in rigid and deformable cavities
- Calculate fundamental frequencies of the cavity, as well as the pressure distribution in the cavity
- Compute the effect of the acoustic medium on the dynamic response of the structure and the influence of the structure on the dynamic response of the acoustic medium
- Cavities may undergo large deformation



Fluid Mechanics

- Perform laminar flow analysis with Navier-Stokes equations
- Conduct studies that involve fluid thermal coupling, fluid-solid coupling and fluid-thermal-solid coupling
- Solve steady state and transient flow problems in two- and three-dimension

Electrostatics and Magnetostatics

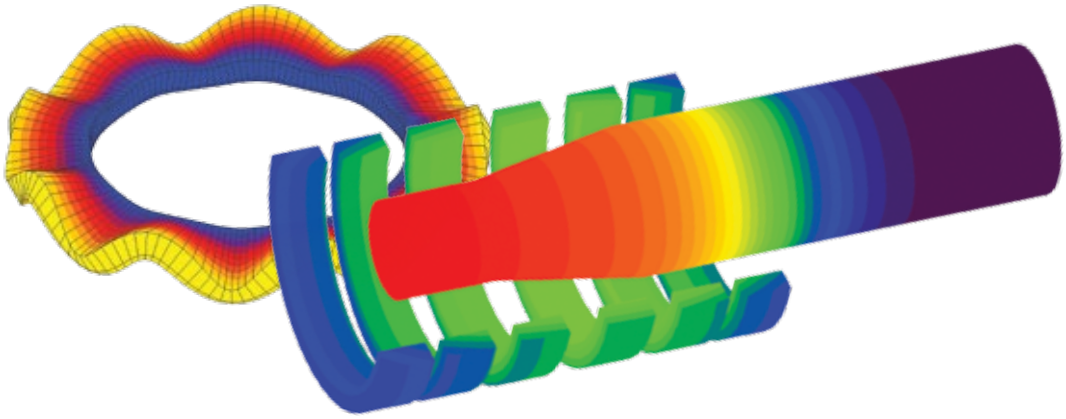
- Evaluate electric fields and magnetic fields in a body or medium
- Compute electric potential field, electric displacement vectors, magnetic induction, magnetic field vector, and more to gain insight
- Model infinite domain with semi-infinite elements for improved accuracy
- Determine the capacitance between electrical conductors in electrostatic analyses
- Calculate the inductance due to wires or coils in magnetostatic analyses

Electromagnetic Analysis

- Perform transient and harmonic fully coupled electromagnetic analysis to calculate electrical and magnetic fields subjected to external excitation
- Compute magnetic permeability as a function of magnetic field in a transient analysis
- Calculate magnetic flux density, magnetic field vector, electric flux density, and electrical field intensity along with potential, nodal charge, and current

Piezoelectric Analysis

- Simulate piezoelectric effect of coupling of stress and electric field in a material
- Solve for nodal displacements and electrical potential simultaneously
- Perform static, transient dynamic, harmonic, and eigenvalue analysis to better understand material response
- Couple with heat transfer analysis to perform a coupled thermal-piezoelectric analysis



Coupled Electrostatic-Structural Analysis

- Simulate MEMS
- Simulate the influence of Coulomb forces on structural components and deformation influence on electrostatic field
- Model contact between different bodies and simulate influence of their interaction on the field

Coupled Thermal-Electrical Analysis (Joule Heating)

- Simulate heaters, defoggers, electronic devices, actuators
- Compute heat generated due to electric flow in a conductor
- Model temperature dependent resistance and internal heat generation as a function of the electrical flow
- Determine the resistance of the device

Coupled Electrical-Thermal-Mechanical Analysis

- Simulate structural response due to Joule heating effects
- Account for nonlinearities that arise due to convection, radiation, and temperaturedependent thermal conductivity and specific heat
- Simulate structures with nonlinearities due to geometric and material behavior
- Use contact analysis to analyze interaction between multiple components

Coupled Magnetostatic-Structural Analysis - Thermal

- Simulate the influence of Lorentz forces on structural components and deformation influence on magnetostatic field
- Model contact between different bodies and simulate influence of their interaction on the field

Coupled Electromagnetic-Thermal Analysis (Induction Heating)

- Simulate induction heating with staggered approach of harmonic electromagnetic analysis followed by thermal analysis
- Compute induced current which generate heat and heat flux
- Incorporate temperature dependency for material data for improved accuracy

Coupled Electromagnetic-Structural -Thermal Analysis

- Simulate induction heating with staggered approach of harmonic electromagnetic analysis followed by thermal-stress analysis
- Compute induced current which generate heat and heat flux
- Incorporate temperature dependency for material data for improved accuracy
- Accurately determine the surface strains and stresses
- Utilize either adaptive meshing or a dualmesh approach to model large motion through the air
- Predict the amount of power required for surface treatment

Material Models for Accurate Results

Simulate materials accurately with the comprehensive library of engineering material models that range from simple linear elastic materials to complex time and temperature-dependent materials, providing you with accurate results.

> "The simulation performed using Marc has given us a good feeling for the behavior of the Ramspol rubber dam and has improved our knowledge of this kind of flexible structures"
>
> Hans Dries, Hollandsche Beton-en Waterbouw

Elastic-Plastic Materials

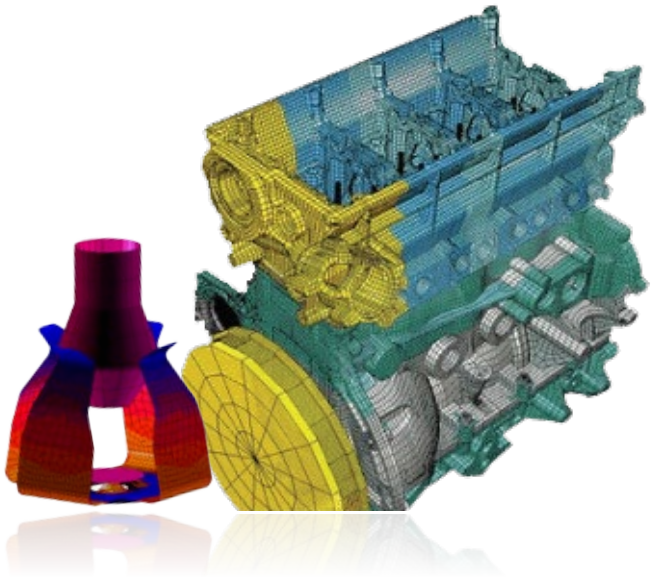
- Model isotropic, orthotropic, and anisotropic elastic materials
- Use isotropic, kinematic, or combined hardening rules to model work hardening
- Model plasticity accurately selecting from a choice of yield criteria, including von Mises, Hill anisotropic yield function, Barlat's, Mohr-Coulomb, and Exponential Cap Model
- Simulate granular material behavior accurately through powder material model
- Study the effects of cyclic plasticity on plastic strains using the Chaboche model
- Model the damage of ductile metals
- Incorporate thermal and rate effects with the Power law models, Johnson-Cook, Cowper-Symmonds, or Kumar models

Time-dependent Inelastic Material Models

- Analyze creep which is an important phenomenon at higher temperature
- Study the combined effects of plasticity and creep on structural behavior
- Simulate the dilatation creep (swelling) on a structure for improved accuracy
- Material data fitting
- Analyze isotropic and anisotropic behavior through thermorheologically simple viscoelastic material model
- Incorporate frequency dependent damping models based upon the viscoelastic nature of rubber and plastics
- Improve result accuracy with material models of choice to model viscoplastic behavior
- Include temperature effects in creep and relaxation studies for improved results

Composite Materials

- Model shell structures with layers composed of different materials or layers of the same anisotropic material with various layer thicknesses and different orientations
- Incorporate both linear and nonlinear material properties in each layer
- Model the degradation of the material using progressive failure analysis (PFA)
- Model the delamination of materials
- Use any of the multiple orientation specification options that provide modeling flexibility



Nonlinear Hypoelastic Material

- Model a generalized nonlinear elastic material to simulate behavior of materials with nonlinear stress-strain relationship
- Use any of the multiple modeling options, which include stress-strain relationship based model, strain invariant model, principal strain space model, bimodulus elasticity which could incorporate no tension, limited tension, no compression, or limited compression

Shape Memory Material

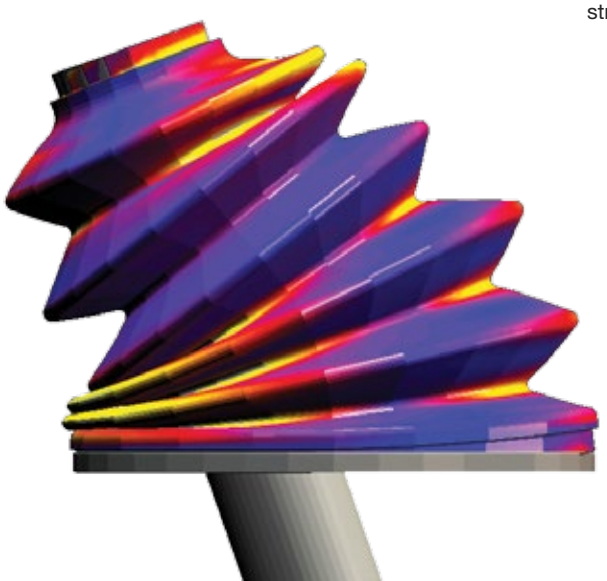
- Simulate the reversible, thermoelastic transformation of shape memory alloys between high temperature austenitic phase and low temperature martensitic phase
- Model transformation induced deformation and irreversible permanent deformation of the martensite
- Choose from mechanical and thermomechanical shape memory models depending on your loading conditions

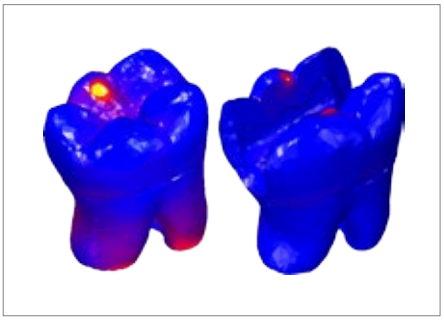
Elastomers

- Choose from multiple material models including Generalized Mooney-Rivlin, Ogden, Arruda-Boyce, Bergrstrom-Boyce, Gent, and Marlow to represent elastomers
- Easily obtain required material parameters for elastomeric material models through integrated curve fitting functionality
- Analyze thermal effects and creep/ relaxation of elastomers through large strain viscoelastic model

Special Class Materials

- Model complex, multi-layer gasket elements that are often made of different materials of varying thickness
- Model complex loading and unloading behavior
- Perform mechanical, thermal, or thermomechanically coupled analyses for desired accuracy and required outputs in a gasket analysis
- Analyze low tension materials like concrete to accurately determine structural strength and behavior under complex loads
- Incorporate rebars to model reinforced structures like composites, biological materials, or reinforced concrete
- Analyze soil material behavior using formulations like Drucker-Prager, Mohr-Coulomb, Cam-Clay, Exponential Cap, and their variations





Easy to use and Powerful Contact Analysis

Utilize Marc's unique contact modeling approach and technology to solve your tough analysis problems that could include large deformation and rotation. Take the hard work out of contact model creation with easy, intuitive set up.

"Marc is a very good product to study both contact and the thermo-mechanical behavior of contacting parts"

Luigi Piccamiglio, Invernizzi Presse

Contact Setup

- Define contact bodies (linear and quadratic) by selecting elements without the need to define contact boundary elements
- Reduce contact set up time through automatic contact boundary detection
- Reduce your modeling time and improve accuracy by using the CAD geometry surfaces and curves to define rigid bodies
- Apply velocities, displacement, loads, moments, and rotations to rigid bodies
- Use rigid bodies to apply symmetry boundary conditions
- Use contact tables to customize contact detection between bodies
- Accurately analyze contact between shells and shells or shells and solids
- Predict contact on both sides of shells
- Analyze beam contact with arbitrary beam cross-sections
- Analyze tube-in-tube contact behavior observed in the oil, automotive, and biomedical industries using beam elements but with 3D contact behavior
- Use segment-to-segment contact method for smoother results contours and to overcome the limitations of master-slave contact method
- Easily model interference fit and overclosure

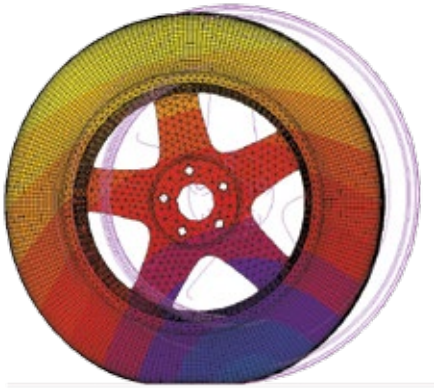
Friction

- Specify different friction coefficients between different contact pairs
- Choose from multiple friction models to suit your requirements – Coulomb friction, shear friction, stick-slip friction, and bilinear friction model
- Use glue option to analyze contact with very high friction by preventing relative tangential motion
- Analyze glue failure between contact bodies by automatic glue deactivation
- Analyze thermally driven contact conditions
- Friction coefficient Near thermal contact may be applied to represent



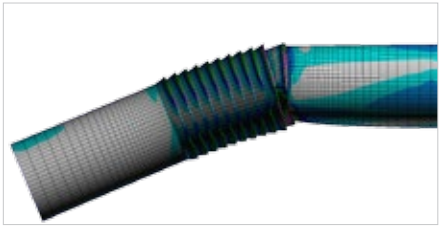
Contact in Coupled Analysis

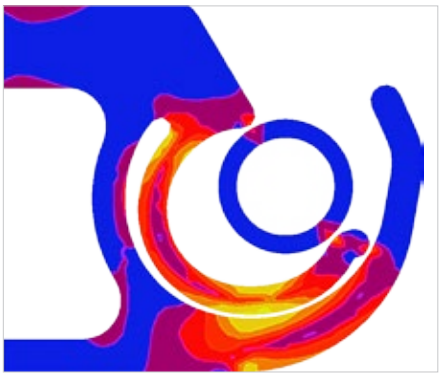
- Perform thermal contact analysis without structural analysis to study heat transfer between different bodies
- Study effects of friction generated heat on structural behavior with support for coupled analysis
- Contact may be used in all multi-physics simulations
- Analyze flow of current between contact bodies for accurate result
- Contact parameters such as the friction is efficient may be function of temperature or other state variables.



Contact Results

- Analyze the progression of contact between contact bodies
- Study the contact forces and pressures and map the distribution in the contact regions
- Sum up the contact forces on a body to evaluate the total forces
- Obtain the total contact area





Automatic Adaptive Meshing for Improved Convergence

Save time and effort by taking advantage of automatic mesh updates in Marc. Get a quality mesh when it is needed most.

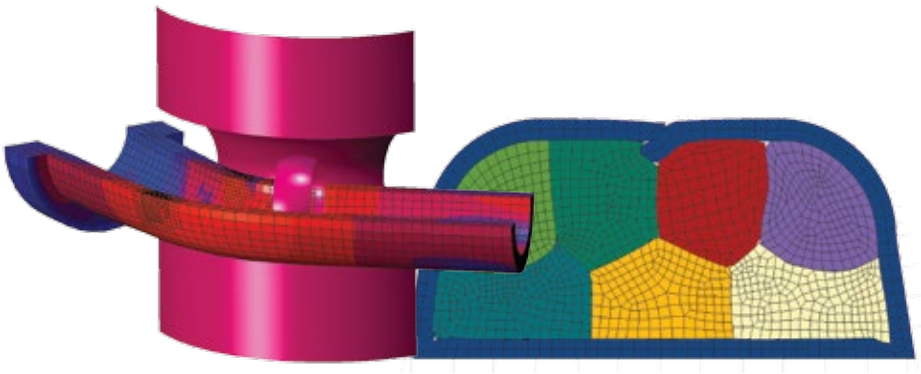
> "(Using Marc) we were able to substitute numerical simulation for the physical development runs, which was very cost-effective"
>
> Charles Courlander,
> Gaurdian Automotive Luxembourg

Local Adaptivity

- Increase the number of elements to be in critical regions to improve accuracy and minimize the cost
- Automatically update the mesh by subdividing elements that satisfy any of several criteria available
- Update boundary conditions on newly created elements and faces
- Attach the nodes to a curve or surface associated with rest of the mesh to smoothen the boundary

Global Adaptivity

- Automatically replace the distorted mesh of a contact body with a better quality mesh
- Map the results of the old mesh to the new mesh without user intervention
- Automatic transfer of boundary conditions to the new mesh
- Solve 2D and 3D models with improved convergence and accuracy
- Generate new refined meshes based upon the solution quantities, such as plastic strain or strain energy density.
- Utilize adaptive meshing to capture crack propagation



Manufacturing Processes

Simulate manufacturing of your product to optimize your manufacturing process and to incorporate the effects of residual stresses in product analysis.

"We were able to compare a virtual video of the deformation generated... with real video footage shot in one of our plants. The match was impressive"

Nicholas Behr, Setforge Engineering

Forming

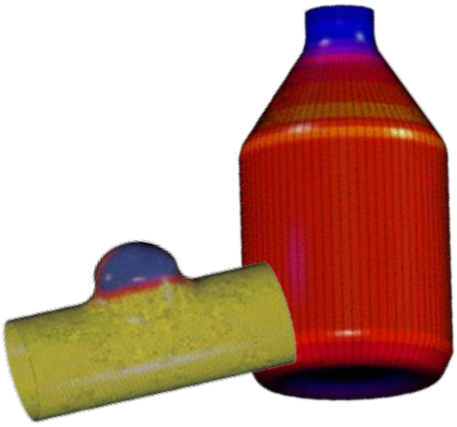
- Simulate forming operations like brake forming, hydroforming, forging, stretch forming, deep drawing, and roll forming
- Cold forming or hot forming operations
- Simulate continuous operations like extrusion and rolling
- Easy model multi-stage processes
- Compute and minimize residual stresses for improved designs
- Accurately calculate the spring-back during the manufacturing process
- Applicable to metals - glass & plastics
- Use automatic remeshing to obtain accurate results and improved convergence

Other Processes

- Simulate welding process and its effects on final structural deformation
- Import NC machine file and analyze the deformations induced by material removal
- Study assembly process with bolt installation and riveting
- Remove the residual stresses with annealing for subsequent product testing

Composites

- Analyze progression of curing and optimize the design and process parameters
- Predict shrinkage caused by curing of resin
- Perform cure-thermal-mechanical coupled analysis to predict residual stress buildup during the curing process



Wear and Failure Analysis

Predict product failures to improve your designs and reduce warranty costs. Reduce development costs by limiting costly destructive testing.

"The capacity in Marc... to automatically internally restart the increment after crack growth, and hence capture multiple crack growth events in one increment, was highly beneficial in terms of computational efficiency"

NASA/CR-2010-216709

Damage Models

- Compute damage accumulation in ductile materials using the Bonora or Gurso growth-based model
- Calculate damage factors using Lemaitre, Cockroft-Latham, or Oyane model
- Simulate Mullin's effect and damage accumulation due to polymer chain breakage in elastomers
- Combine rubber damage model with viscoelastic behavior using the Bergstrom-Boyce or parallel model
- Study brittle cracking and crushing in concrete

Composite Failure

- Analyze laminate failure by choosing from industry accepted failure criteria that include maximum stress and strain criteria, Hill, Hoffman, Tsai-Wu, Hashin, Hashin Fabric, Hashin tape, Puck, and Strain Invariant Failure Theory (SIFT) failure criteria
- Analyze progressive failure of composites
- Study delamination failure using Cohesive Zone Modeling of the bonding between lamina

Fracture Mechanics

- Evaluate energy release rate and stress intensity factor using Lorenzi method or Virtual Crack Closure Technique (VCCT)
- Compute energy release rates for brittle and ductile materials at small and large strains

Automatic Crack Propagation

- Analyze crack growth under direct loading Rheological model or fatigue loading
- Simulate crack growth along edges or along contact region
- Simulate crack propagation to accurate track the crack front or utilizing automatic adaptive meshing
- Determine the low cycle or high cycle fatigue life

Wear

- Analyze mechanical erosion due to impact of particles, part vibration, or shear stress due to flow
- Study erosion due to pyrolysis involving decomposition of material due to thermal process
- Model thermochemical ablation of material subjected to high thermal fluxes
- Study tool effectiveness by simulating tool wear

Efficient Solvers and Parallel Processing for Higher Productivity

Take advantage of efficient matrix solvers and unique parallel processing methodologies to get results faster.

"I am very impressed with the capabilities... I'm very satisfied with the Pardiso Solver and the speed increase it provides"

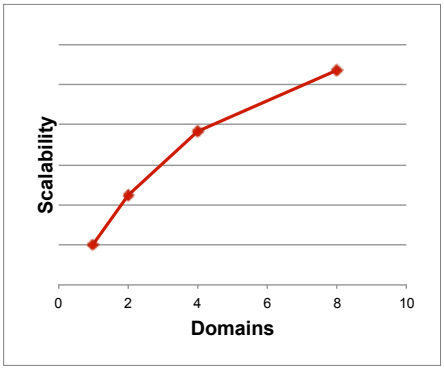
Giuseppe Simonetti, BogWarner Inc.

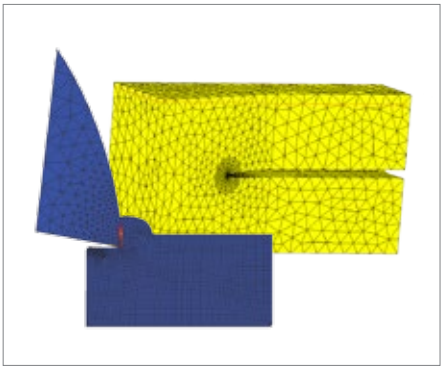
Solvers

- Use solvers designed for best performance, including multifrontal, Pardiso, MUMPS, and CASI iterative solver
- Use iterative solvers to run larger models without high memory usage
- Use out-of-core option to make efficient use of disk space to solve large models
- Benefit from NVIDIA™ GPU for improved performance
- Solver specific bandwidth optimization to reduce storage space

Parallelization

- Speed up your solution by using all available cores on multicore desktops
- Obtain true parallelization through Domain Decomposition Method by parallelizing all phases of the solution - namely input, assembly, solution, stress recovery, and output
- Automatic and manual decomposition of the model for parallelization
- Make the best use of your hardware - whether a shared memory parallel or distributed memory parallel system







MSC Software is one of the ten original software companies and the worldwide leader in multidiscipline simulation. As a trusted partner, MSC Software helps companies improve quality, save time and reduce costs associated with design and test of manufactured products. Academic institutions, researchers, and students employ MSC technology to expand individual knowledge as well as expand the horizon of simulation. MSC Software employs professionals in 20 countries.

For additional information about MSC Software's products and services, please visit: ***www.mscsoftware.com.***

©2017 Hexagon AB and/or its subsidiaries and affiliates. MSC Software is part of **Hexagon**. The MSC Software corporate logo, MSC, and the names of the MSC Software products and services referenced herein are trademarks or registered trademarks of the MSC Software Corporation and/or its subsidiaries and affiliates in the United States and/or other countries. All other trademarks belong to their respective owners. All rights reserved.

MARC*2017OCT*BR