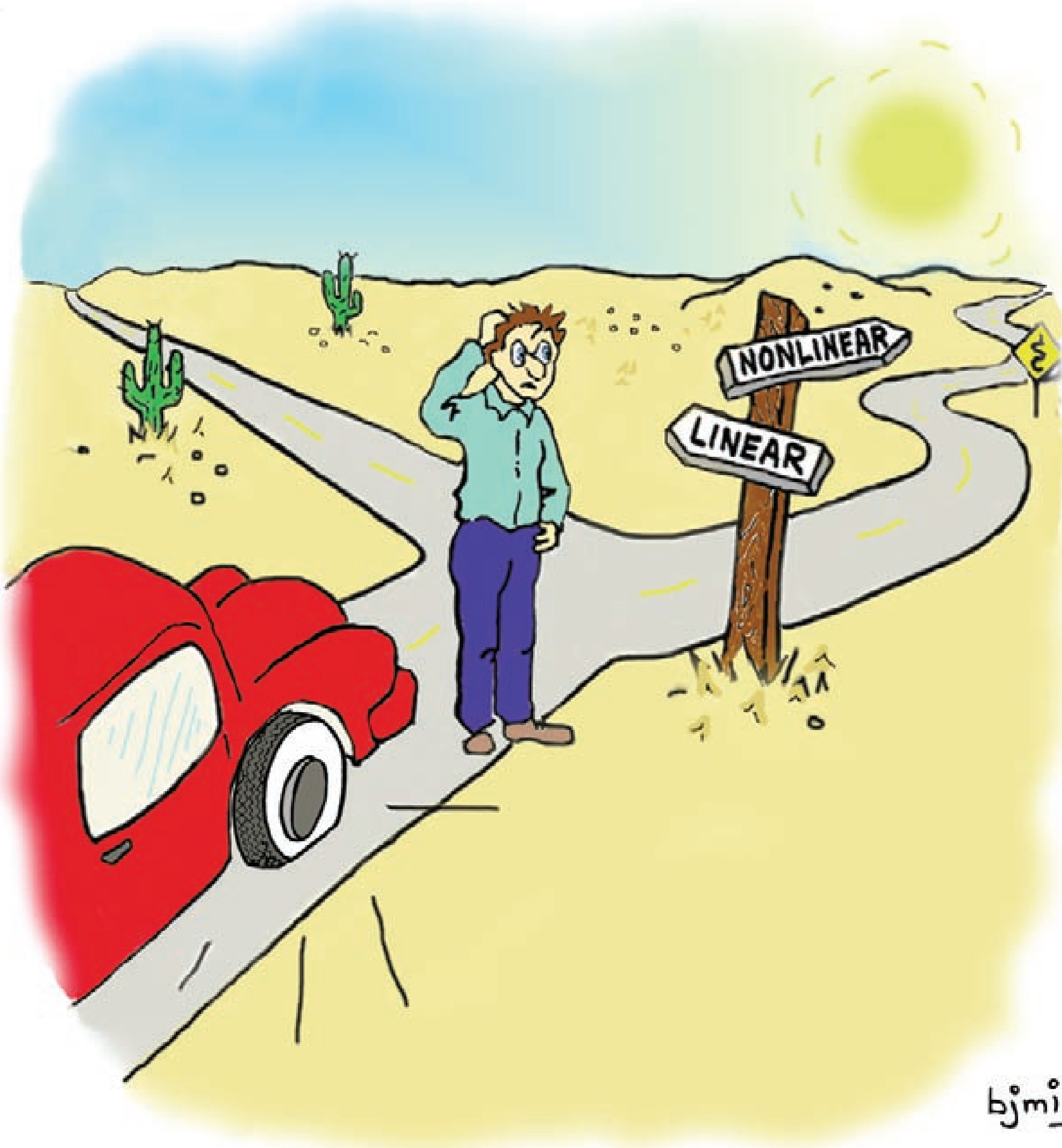
"When does $F \neq Ku$?"

MSC.visualNastran

Technical White Paper



MSC SOFTWARE
SIMULATING REALITY

Let us take you on a Nonlinear Journey

- Introduction 1
- How do I spot a nonlinear problem? 2
- When should I use nonlinear? 3
- What is the solution procedure? 4
- What are material nonlinearities? 5-6
- What are geometric nonlinearities? 7
- How do I model boundaries? - Contact! 8
- What about dynamics? 9
- What are the benefits of FEA? 10
- The world is full of nonlinear problems! 11-12
- What else does MSC.Software offer? 13

Introduction

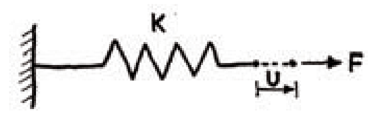
Products that don't perform like they should can be very inconvenient, as depicted by our dedicated engineer on the cover. Although fixing the flat tire is his immediate concern, avoiding the situation altogether through improved product design is smarter. Our dedicated engineer ponders the design and analysis considerations of modeling advanced products like a tire, hoping to never be stuck without knowing which way to go. Join our engineer on his journey as he searches for a fix for his tire problem, and discovers that $F \neq Ku$ is typically the rule, not the exception.

F = Ku

Analysis in structural or mechanical engineering means the application of an acceptable analytical procedure based on engineering principles. Analysis is used to verify the structural or thermal integrity of a design. Sometimes, this can be done using handbook formulas for simple structures. More often, however, this analysis is performed using computers in order to predict product performance. The pre-dominant type of engineering software used in these analyses is based on the finite element method, and this type of analysis is therefore called finite element analysis (FEA).

In the past 40 years, FEA has been successfully applied in all major industries, including: **aerospace, automotive, energy, manufacturing, chemical, electronics, consumer, and medical industries**. FEA is indeed one of the major breakthroughs of modern engineering.

The origins of mechanics go back to early scientists such as Isaac Newton and Robert Hooke. All freshmen physics students learn Hooke's Law, as illustrated by simple spring, with stiffness K (N/m) loaded at the free end by a force F (N):



Hooke discovered a simple linear relationship between force and deflection, $F = Ku$.

Thus, the deflection u can be easily calculated by dividing F by K, (This law is valid as long as the spring remains linear elastic, and the deflections are such that they do not cause the spring to yield or break.) If one applies twice the force, the spring will deflect twice as much. Simple - but not all problems exhibit this simple behavior, for example, our dedicated engineer's tire.

FEA History

In the mid-fifties, American and British aeronautical engineers independently developed the finite element method to analyze airplane structures.

"How do I spot a nonlinear problem?"

A structure can be idealized as composed of many small, discrete pieces called finite elements. These engineers extended Hooke's basic idea into large structures involving thousands of simultaneous equations, and were able to solve these equations using the first generation of computers.

Since linear FEA theory was formulated first, these early structural analyses were linear. In the sixties, researchers started applying the finite element method to other fields in engineering science, such as fluid mechanics, heat transfer, electromagnetic wave propagation, and other field problems. Applied mathematicians proved that the method converges to the correct results. Researchers also started applying FEA to nonlinear problems.

How to Spot a Nonlinear Problem

Some everyday situations are nonlinear, and you probably don't even recognize them: pressing your finger against a balloon to see a dimple; flexing a paper clip back and forth; crushing an aluminum can so that it buckles; and hitting a pothole in the road. These cases all exhibit large deformations, and sometimes, inelastic material behavior.

The three major types of nonlinearities are as follows:

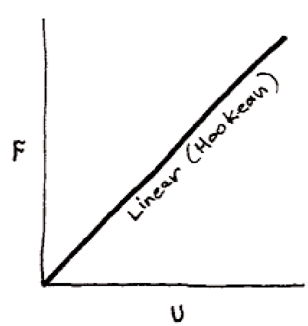
- **Material Nonlinearity** (plasticity, creep, viscoelasticity)

- **Geometric Nonlinearity** (large deformations, large strains, snap-through buckling)

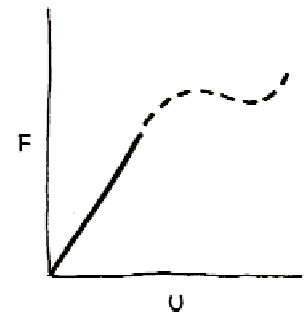
- **Boundary Nonlinearity** (opening/closing of gaps, contact, follower force)

You can, of course, have combinations of any of these. To find evidence of possible nonlinear behavior, look for: permanent deformations and any gross changes in geometry, cracks, necking, thinning, distortions in open section beams, crippling, buckling, stress values which exceed the elastic limits of the materials, evidence of local yielding, shear bands, and temperatures above 30% of the melting temperature. In these cases, the stress is no longer proportional to the strain.

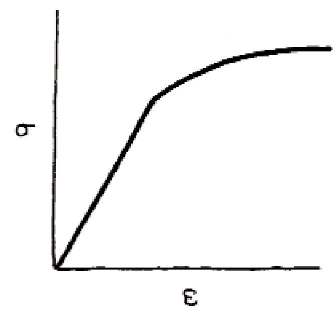
Typical force-displacement curves are shown for linear (Hookean) and nonlinear material, as well as nonlinear stress-strain curve:



Linear Force-Displacement



Nonlinear Force-Displacement



Nonlinear Stress-Strain

How about our Tire?

Tires are complex products that have high performance demands. They must have adequate traction, and acceptable wear.

“When should I use nonlinear?”

They should have low rolling resistance, for fuel economy. They must not debond or delaminate. They should roll quietly. They should not hydroplane under wet conditions, and ultimately, they should not blowout or get flat, leaving you stranded in the middle of the desert!

Effects due to friction, temperature and dynamics are important, which is why major tire manufacturers around the world use nonlinear FEA capabilities to improve tire designs, ride comfort and service life. A tire analysis can involve every type of nonlinearity.

To properly design a tire, a realistic model must be created, and designs optimized. The more designs you can turn around and optimize in a virtual environment, the more you will save on costly physical prototyping and testing. So what's involved? Let's look at nonlinear problems in general first.

Nonlinear FEA Concepts

You should recognize at the outset that nonlinear problems are inherently more complex to analyze than linear problems. And, the "principle of superposition" (which states that the resultant deflection, stress, or strain in a system due to several forces is the algebraic sum of their effects when separately applied) no longer applies.

Finite element analysis is an approximate analysis method which is only as accurate as:

- *the quality of the model*
- *the material properties used (and their assumptions)*
- *representation of the loads and boundary conditions*
- *the solution algorithm.*

The analyst's experience and judgement therefore become critical to the success of a nonlinear analysis because of the decisions that must be made. In nonlinear FEA, the following relationships (which are assumed to be linear in linear FEA) may be violated:

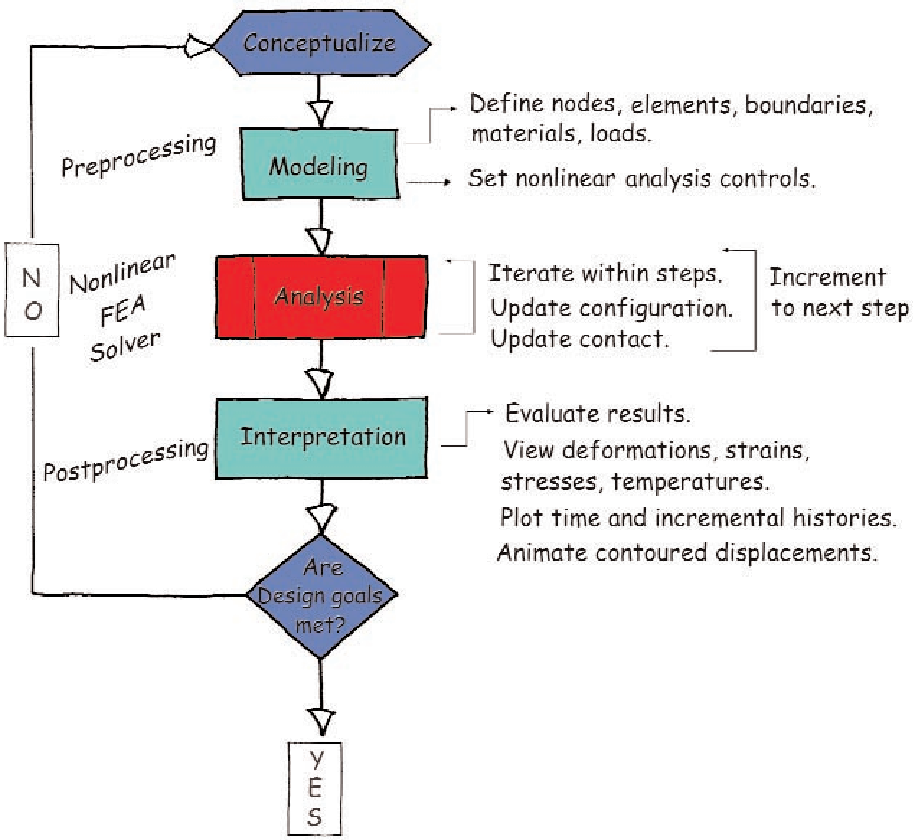
1. The strain is no longer small.

Most metallic materials are no longer useful when the strain exceeds one or two percent. However, some materials, notably rubbers, elastomers, and plastics, can be strained to hundreds of a percent and will therefore require finite (large) strain analysis.

2. The strain-displacement relationship is no longer linear.

This is true if the rotations become large even though the strains are still small. The changes in the deformed shape can no longer be ignored. The physics of buckling, rubber analysis, metal forming, among others, requires that either a quadratic relationship exits between the strain and displacement (Green-Strain) or a logarithmic relationship exists. Engineering stress is no longer appropriate because of geometric changes and the true stress or Cauchy stress should be used.

"What is the solution procedure?"



3. The stress-strain law may become nonlinear.

Even within the useful stress range of the material. This behavior is typical of most metals, rubbers and elastomers, and certain composite materials whose properties are unequal in tension and compression.

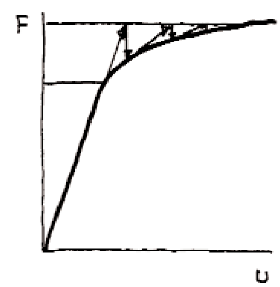
4. The original equilibrium equations (relating stress to loads) may have to be updated.

Due to the geometrical changes in the shape of the structure. These relations mean that, in nonlinear FEA, the load is no longer proportional to the displacement, that is, $F \neq Ku$.

Incremental Solution Procedures

FEA is an approximate technique, and there exist many methods to solve the basic equations. In nonlinear FEA, two popular incremental equilibrium equations are: full Newton-Raphson and modified Newton-Raphson.

The **full Newton-Raphson** (N-R) method assembles and solves the stiffness matrix every iteration. It has quadratic convergence properties, which means in subsequent iterations the relative error decreases quadratically. It gives good results for most nonlinear problems.



In addition, other solution procedures offered in nonlinear FEA codes include:

- **Modified Newton Raphson**
- **Strain Correction Method**
- **Secant Method**
- **Direct Substitution Method**
- **Quasi-Newton Methods**

"What are material nonlinearities?"

When stresses go beyond the linear elastic range, material behavior can be broadly divided into two classes:

- **Time-independent behavior** (plasticity-applicable to most ductile metals; nonlinear elasticity-applicable to rubber, elastomers)

- **Time-dependent behavior** (creep, viscoplasticity-applicable to high-temperature applications, concrete; viscoelasticity-applicable to elastomers, glass, plastics).

Here we'll give you a brief introduction to these concepts (many of which are quite complex and are the subjects of textbooks). An elastic-plastic material may be defined as a material, which, upon reaching a certain stress state, undergoes deformation, which is irreversible. This results in a behavior, which is path-dependent. A basic assumption in elastic-plastic analysis is that deformation can be divided into an elastic part and in an elastic (plastic) part.

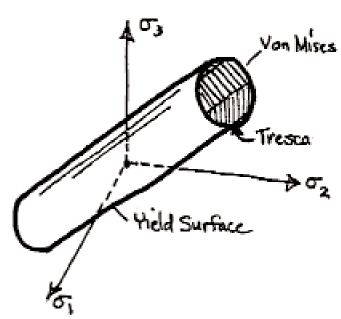
Yield Conditions

The *yield stress* is a measured value that separates the elastic and inelastic behavior of a material. Its magnitude is usually obtained from a uniaxial test. However, stresses in a structure are multiaxial, and a measure of yielding in a multiaxial state of stress is called the "yield condition" or "yield criterion."

The most widely used yield condition is the *Von Mises*, which states that yielding occurs when effective (or equivalent) stress equals the yield stress as measured in a uniaxial test. This yield condition agrees fairly well with the observed behavior of ductile metals such as low-carbon steels and aluminum.

Another yield condition is the *Tresca*, which states that yielding occurs when the maximum shear stress reaches the value it has when yielding occurs in the tensile test.

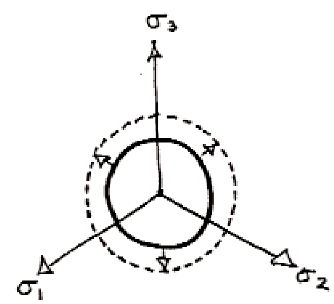
A third yield condition is the *Drucker-Prager* (or *Mohr-Coulomb*), which is based on a yield surface that exhibits hydrostatic stress dependence. Such behavior is observed in materials such as certain soils, rock-like materials, and ice.



Many advanced yield criteria have been formulated either for specific materials or observed phenomena. This includes Gurson model for damage, Shima for powder materials, and Cam-Clay for soils to name but a few.

Work Hardening Rules

In a uniaxial test, the work hardening slope is defined as the slope of the stress-plastic strain curve. It relates the incremental stress to incremental plastic strain in the inelastic region and dictates the conditions of subsequent yielding.

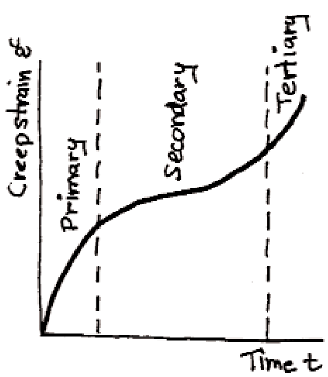


The *isotropic* hardening rule assumes that the center of the yield surface remains stationary in the stress space, but the size (radius) of the yield surface expands due to strain hardening. It is considered suitable for problems in which the plastic straining far exceeds the incipient yield state. It is therefore widely used for manufacturing processes and large-motion dynamic problems. The kinematic-hardening rule assumes that the Von Mises yield surface does not change in size or shape, but the center of the yield surface shifts in stress space.

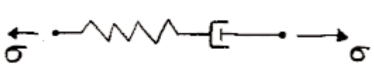
Straining in one direction reduces the yield stress in the opposite direction. It is used in cases where it is important to model the Bauschinger effect.

Creep

Creep is continued deformation under constant load, and is a type of time-dependent inelastic behavior, which can occur at any stress level. It is an important consideration for elevated-temperature stress analysis (e.g., in nuclear reactors) and in materials such as concrete. Creep behavior can be characterized in three stages: primary, secondary, and tertiary.



Creep is generally represented by a *Maxwell model*, which consists of a spring and a viscous dashpot in series. For materials that undergo creep, with the passing of time the load decrease for a constant deformation. This phenomenon is termed *relaxation*.

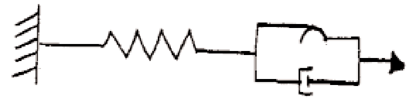


Viscoelasticity and Viscoplasticity

Viscoelasticity, as its name implies, is a generalization of elasticity and viscosity. It is often represented by a *Kelvin model*, which assumes a spring and dashpot in parallel. Examples of viscoelastic materials are glass and plastics.

A *viscoplastic* material exhibits the effects of both creep and plasticity, and can be represented by a creep (Maxwell) model modified to include a plastic element.

Such a material behaves like a viscous fluid when it is in the plastic state.

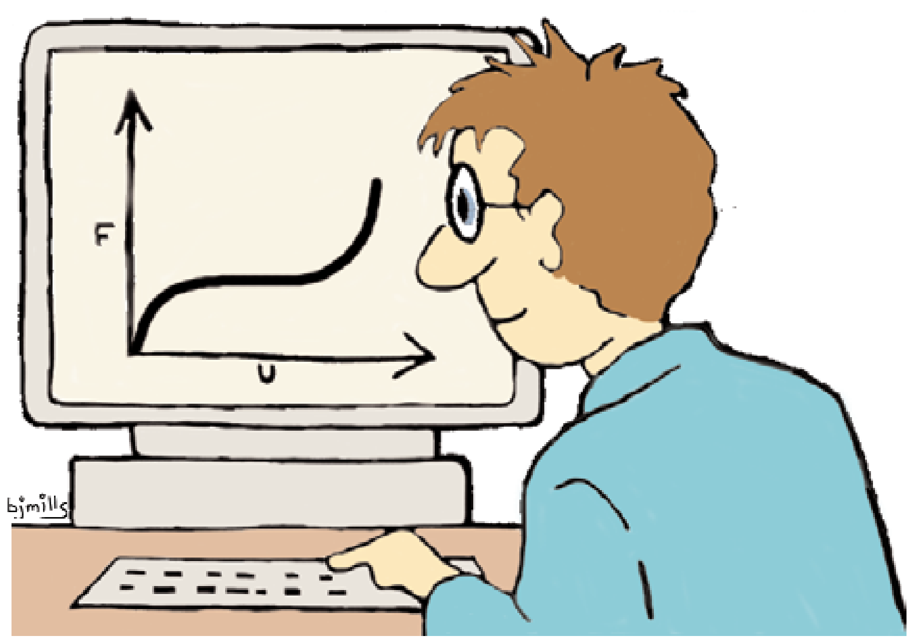


Rubber and Elastomers

An elastomer is a natural or synthetic polymeric material with rubberlike properties of high extensibility and flexibility. Elastomers show nonlinear elastic stress-strain behavior, which means that upon unloading the stress-strain curve is retraced and there is no permanent deformation.

Tire Material Modeling

Tires can use the generalized Mooney, Ogden, Arruda-Boyce and Gent models, or you can define your own model through user-routines.

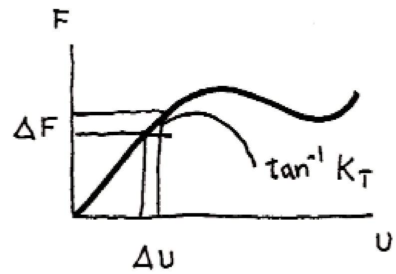


"What are geometric nonlinearities?"

Rubber and elastomers are nearly incompressible, which means zero volumetric change under load and a Poisson's ratio of nearly equal to one-half. This incompressibility constraint means that FEA codes can handle these types of materials only if they have certain special element formulations (for instance, "Herrmann elements" or "mixed/hybrid formulations"). FEA codes with this nonlinear elastic capability generally offer one or more hyper-elastic material models (strain energy functions), such as the two-constant *Mooney-Rivlin* model, one-constant Neo-Hookean model, and the five-constant, third-order James-Green-Simpson model. Material constant for these models are obtained from experimental data.

Large Deformations

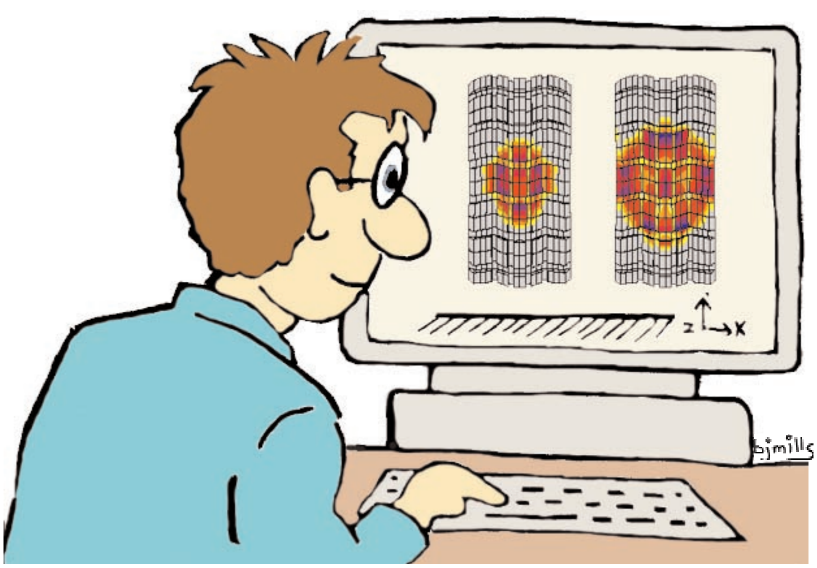
In implicit analysis, using the Newton Raphson procedure, the relationship between incremental load ΔF and displacement Δu is called a tangent stiffness K_T.



This stiffness has three components: the material stiffness, the initial stress stiffness, and the geometric stiffness. The material stiffness may be the same elastic stiffness as that used in linear FEA.

Tire Deformations

Tires can be modeled using a large strain viscoelasticity model with damage effects, such as the Mullins and Miehe models.



The second term represents the resistance to load caused by realigning the current internal stresses when displacements occur. The third term represents the additional stiffness due to the nonlinear strain-displacement relationship.

In solving this type of problem, the load is increased in small increments, the incremental displacement Δu is found and the next value of the tangent stiffness is found, and so on. There are three approaches available to solve these types of problems:

· **Total Lagrangian method** which refers everything to the original undeformed geometry, and is applicable to problems exhibiting large deflections and large rotations (but with small strains), such as thermal stress, creep, and civil engineering structures. It is also used in rubber analysis where large elastic strains are possible.

· **Updated Lagrangian method** where the mesh coordinates are updated after each increment, and is applied to problems featuring large inelastic strains such as metal forming.

· **Eulerian method** where the mesh is fixed in space and the material flows through the mesh, and is suitable for steady-state problems such as extrusion and fluid mechanics problems.

“How do I model boundaries? - Contact!”

Contact, by nature, is a nonlinear boundary value problem. During contact, mechanical loads and perhaps heat are transmitted across the area of contact. If friction is present, shear forces are also transmitted. Contact can be achieved in various ways. One way is in MSC.Marc. In MSC.Marc, areas of potential contact do not need to be known prior to the analysis. If the code used interface elements instead, you would have to know where all contact will occur in advance.

Both *deformable-to-rigid* and *deformable-to-deformable* contact situations are allowed in MSC.Marc. You need only to identify bodies which are *potential* candidates for contact during the analysis. Self-contact, common in rubber problems, is also permitted. The bodies can be either rigid or deformable, and the algorithm tracks *variable contact* conditions *automatically.* Rigid surfaces may be directly imported from a CAD system, and their exact mathematical form is used in the calculations. For a deformable body, the geometry is normally represented by the edges of the element, but MSC.Marc can improve upon this by fitting a curve or surface through the boundary. This improves the accuracy of the solution by representing the geometry better than the discrete finite elements. This is important for concentric shafts or rolling simulation.

Using MSC.Marc, you no longer need to worry about the location and open/close status checks of “gap elements,” or about “master-slave” relationships. Also, *coupled* thermo-mechanical contact problems (for example, rolling, casting, extrusion, car tire) and dynamic contact problems can be treated.

Friction

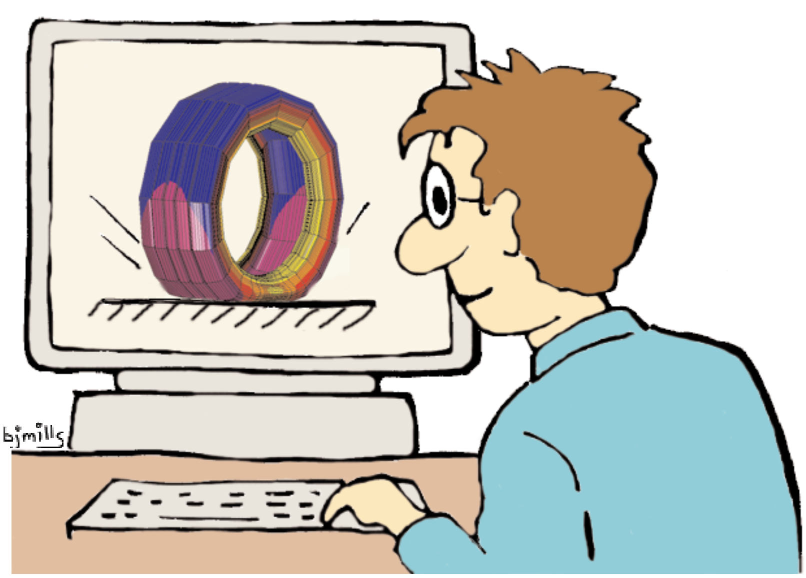
MSC.Marc offers two classical friction models: *Coulomb friction* and *shear friction.* In addition, a *user subroutine* is available in MSC.Marc, permitting you to constantly monitor the interface conditions and modify the friction effect if necessary.

In this way, friction can be made to vary arbitrarily- as a function of location, pressure, temperature, amount of sliding, and other variables. In order to reduce numerical instabilities in the transition between *sticking* and *slipping,* a regularization procedure is applied.

Sometimes, the physics of deformation dictates modeling the regions of sticking fairly accurately (for example, driver pulley transferring torque through the belt to a driven pulley). For such cases, a stick-slip friction model based on Coulomb is also available. Because friction generates heat, a coupled thermo-mechanical analysis is often required in rubber contact problems.

Tire Contact

Frictional contact is used to model the behavior at the footprint using Coulomb and shear friction models. The road surface can be represented as either rigid, or discretized into an FEA mesh. The rim can be modeled with contact as well.



"What about dynamics?"

Nonlinear Dynamic Analysis

An important FEA application is in the area of nonlinear dynamics, for example, in pipe whip, impact, and other intermittent contact problems. To solve the matrix equations of motion numerically, most codes offer either implicit direction integration schemes or explicit schemes.

Implicit

An *implicit* operator solves a matrix system, one or more times per step, to advance the solution. It is appealing for a linear transient problem, because it allows a larger time step and has almost no numerical stability problems. Treatment of boundary nonlinearities must occur within a step, and this fact along with the solution of large systems of equations make the coding more complicated than an explicit one.
Examples of implicit operators include: Newmark-beta (the most popular), Wilson-theta, Hilber-Hughes-Taylor, and Houbolt. Some of these operators have been shown by researchers to exhibit numerical damping problems, sensitivity to time step size, or stability problems, and the user must be extremely careful in their use.

Explicit

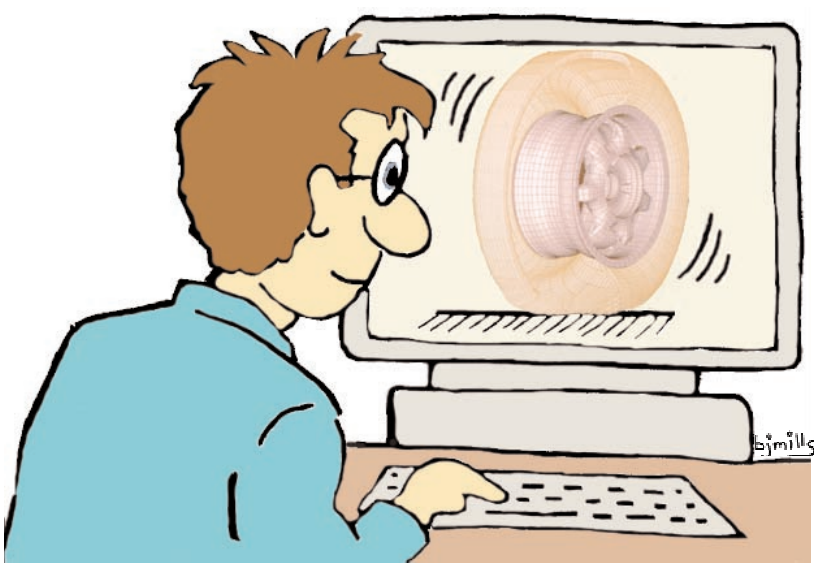
An *explicit* operator advances a solution without forming a stiffness matrix, a fact that makes the coding much simpler. An example of an explicit scheme is the central difference operator, which is conditionally stable. For a given time step size, an explicit formulation requires fewer computations per time step than an implicit one. Complicated boundary conditions are handled easily, because nonlinearities are handled after a step has been taken. The disadvantage of an explicit method is the existence of a definite stability limit, which means very small time steps are required and often, higher computer costs are incurred.

Implicit vs Explicit

The choice of whether to use implicit or explicit time integrationv schemes is very subtle and depends on: the nature of the dynamic problem; the type of finite elements which make up the model; the size of the model; and the velocities of the problem compared to the speed of sound in the material.

Tire Dynamics

Noise due to sidewall vibrations and standing waves require a coupled structural acoustics transient analysis. FEA can handle this as well as thermo-mechanical coupling of either steady state or transient analyses.



"What are the benefits of FEA?"

Virtual Manufacturing

Virtual Manufacturing involves the use of a computer to simulate a product and the processes involved in its fabrication. Simulation technology enables companies to optimize key factors directly affecting the profitability of their manufactured products. These include manufacturability, final shape, residual stress levels, and product durability. Profitability is improved by reducing costs of production, material usage, and warranty liabilities.

At the core of Virtual Manufacturing lies nonlinear FEA technology. The technology has enabled companies to simulate fabrication and testing in a more realistic manner than ever before.

Virtual Manufacturing reduces the cost of tooling, eliminates the need for multiple physical prototypes, resulting in reduced material waste. Because you can "get it right the first time" it provides manufacturers with the confidence of knowing that they can deliver quality products to market on time and within budget.

From a business perspective, it is clear. Small improvements in manufacturing have dramatic and profound effects in terms of cost and quality.

Return on Investment calculations have shown that small savings in material usage deliver enormous returns in a manufacturing environment. For example, an automotive customer found that each ounce of material saved in a forged car engine component saves many hundreds of thousands of dollars of material costs each year. Calculations are simple thanks to the large manufacturing runs, however the same customer went on to calculate the impact on customer satisfaction from the extra power available to the engine and reduced running costs of the final vehicle.

Nonlinear FEA allows you to simulate the behavior of your product and based on the information obtained from the simulation, apply engineering judgment to optimize the design. To summarize, some of the advantages of this enhanced design process achieved:

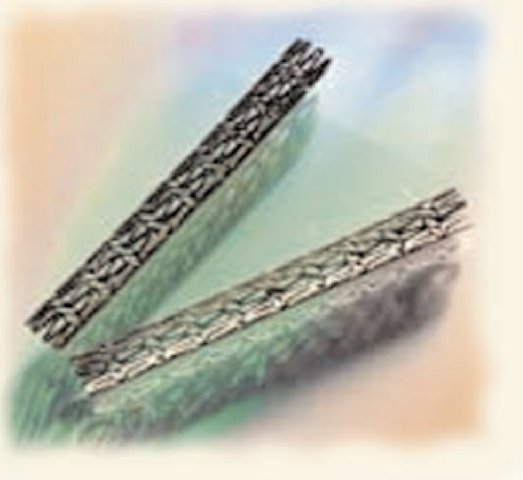
- *Improved performance and quality of finished product*
- *Verification of designs before prototyping*
- *Elimination of costly manufacturing iterations*
- *Reduced material waste*
- *Reduced time-to-market*
- *Competitive advantage over Competitors*

Improving Your Competitive Advantage

The design of products typically is carried out in a trial and error fashion, relying heavily on manufacturing experience, as well as costly shop floor trials. A viable alternative for reducing these design costs and increasing your competitiveness is the use of Virtual Manufacturing.

The primary benefit of Virtual Manufacturing is reduced product development and manufacturing costs, achieved by improved designs. Using computer simulation, designers can quickly eliminate faulty designs and optimize the design before manufacture.

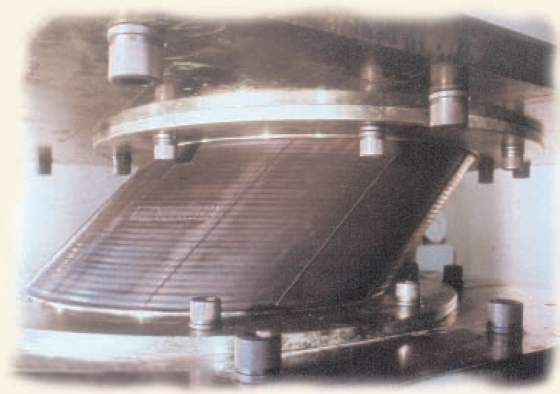
While such Virtual Prototyping techniques and processes are still evolving, it is by no means merely a concept or theoretical construct; it is already a dawning reality and is fully implemented in the form of MSC.Marc, MSC.SuperForm, MSC.SuperForge, and MSC.Dytran.



Biomedical Stents

Head Gasket

Disc Brake



Isolation Bearing

Our enlightened engineer clearly sees many problems requiring nonlinear solutions.

"The world is full of nonlinear problems!"

Biomedical Stent

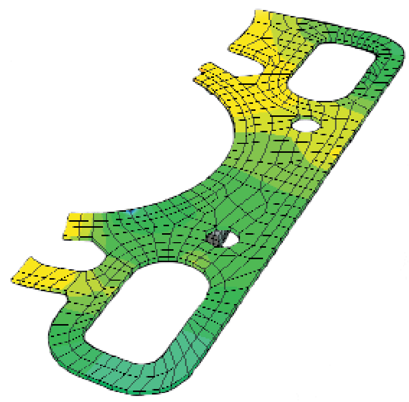
Stents are used at various locations throughout the human body, but some of the most intensive work involves coronary arteries, where stents are used to maintain the opening in the vessels supplying blood directly to the heart muscle.

"Biomedical applications represent some of the most demanding of all work done in FEA. Projects often require modeling of systems consisting of multiple components with nonlinear materials, complex 3D geometries and surface-to-surface contacts as well as coupled conditions that may involve simultaneous mechanical thermal, electromagnetic loading and fluid-structure interaction." (Dr. Svenn Borgersen, BIOSMulations Inc., Eagan, MN)

Biomedical applications have high safety requirements and thus require extensive nonlinear analyses.

Head Gasket

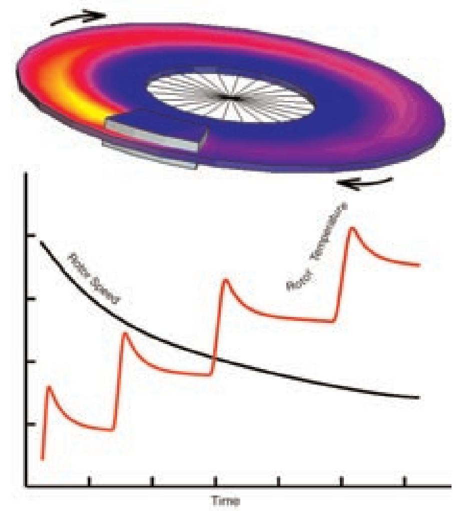
The manufacture of an engine gasket includes a number of nonlinearities. A FEA simulation of tightening the head bolts deforms the gasket's head around the cylinders into the plastic range. The simulation lets you visualize the deformation of the head and determine how well the seal is made.



This analysis improved the gasket's wear, performance, and warranty costs.

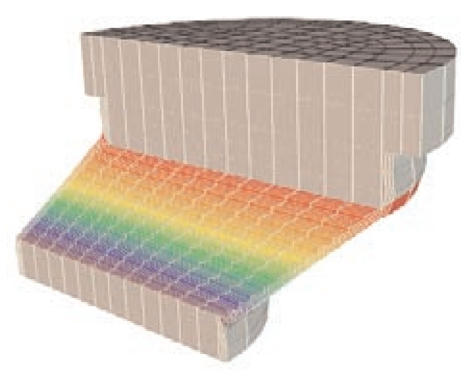
Disc Brake

A nonlinear FEA simulation determines the temperature distribution with time as the brake pads are applied to the disc. A contour plot of temperature on the disc displays the peak temperatures and lets you verify the disc's performance.



Isolation Bearing

Seismic Isolation Bearings are used for earthquake protection of structures. They are composed of an elastomeric compound layered between thin steel shims. Nonlinear \FEA verifies the bearings axial and shear stiffness, as well as the internal stresses in the material under earthquake loading.



These are just a few of the many opportunities you will find to use nonlinear FEA. Remember, "Nature is Nonlinear."

“What else does MSC.Software offer?”

MSC.Software

MSC.Software is the world leader in nonlinear and coupled physics simulation. Our technology has evolved and matured through constant development and worldwide use by thousands of engineers. Our manufacturing tools have helped users save hundreds of millions of dollars since inception, and to this day, continue to catapult profits and slash manufacturing costs. To find out how MSC.Software's many capabilities will enable you to solve your most difficult engineering challenges, contact us online at www.mscsoftware.com or by telephone at (800) 642-7437 ext. 2500.

MSC.visualNastran Enterprise

MSC.visualNastran Enterprise products are built for professional analysts and engineers who need direct access to the full power of MSC.Software's simulation technology. The family includes MSC.Nastran, MSC.Patran, MSC.Marc, MSC.Dytran, MSC.Mvision and MSC.Fatigue, as well as MSC.Software’s renowned industry specific applications. This software is highly functional and is able to be integrated with a wide range of other enterprise software including CAD, PDM, test software and other CAE software.

MSC.Marc

MSC.Marc is an advanced finite element system focused on nonlinear design and analysis as well as the process modeling community. In addition to a unique ability to solve very large problems in parallel using the domain decomposition technique, MSC.Marc is known for great depth in solution procedures, material models, and element technology. Whether your design is of steel, rubber, plastic, glass, or concrete, MSC.Marc can be used to solve the problem.

MSC.SuperForm

MSC.Marc SuperForm is a comprehensive manufacturing process simulation program based on the MSC.Marc technology. It is used by engineering analysts to simulate a wide variety of forming operations, including forging, bending, extrusion, rolling and cogging.

MSC.Dytran

MSC.Dytran is an advanced finite element program capable of simulating many common forming processes, including the forming of complex sheet metal parts such as automobile hood, fenders, and side panels, as well as forming of household and industrial containers like plastic bottles.

MSC.SuperForge

MSC.SuperForge is a Windows based application custom developed for forging process simulation. Its object oriented process modeling, forging process terminology, and "meshless" technology make it an ideal simulation tool for forging process design engineers and shop floor technicians.

MSC.visualNastran Desktop

MSC.visualNastran Desktop products have been developed to support the many design engineers at large or small companies who need to verify the functionality of their design concepts without extensive simulation support. This family includes the popular CAD integrated versions of MSC.visualNastran 4D, the leading PC based motion simulation package. Program management as well as Marketing professionals will benefit from sophisticated simulation work through the use of MSC.visualNastran Studio. Highly advanced visualization and animation capabilities provide the insight required to make design intuitive.

"Thanks MSC.Software!"

Brochure produced by Marlene Hoctor and Ludi Billings

MSC.Software provides the industry's most comprehensive support system with over 50 offices worldwide to provide local and centralized support. Investing in MSC.Software gives you access to extensive client support through comprehensive documentation, direct technical expertise, and customized training classes.

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
1 714 540.8900
Fax: 1 714 784.4056

Information Center:

1 800 642.7437 ext. 2500 (U.S. only)
1 978 453.5310 ext. 2500 (International)

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

Europe:

MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
49 89 43 19 87 0
Fax: 49 89 43 61 71 6

Asia-Pacific:

MSC Japan Ltd.
Entsuji-Gadelius Bldg.
2-39,Akasaka 5-chome
Minato-ku, Tokyo 107-0052 Japan
81 3 3505 0266
Fax: 81 3 3505 0914

MSC SOFTWARE
SIMULATING REALITY