4. SOLVING PHASE

The next step in the FEA analysis is to send the model off and let the computer do all the calculation work. The software that does all the calculations is called the **solver** and it goes through the meshed model you've created and solves a bunch of mathematical equations for each of the nodes to figure out overall stress and deformation of the part.

These equations are based on the old **F=kx** equation for a spring, where **F** is the force, **k** is the spring stiffness and **x** is the displacement of the spring, or how much it stretches due to the applied force.

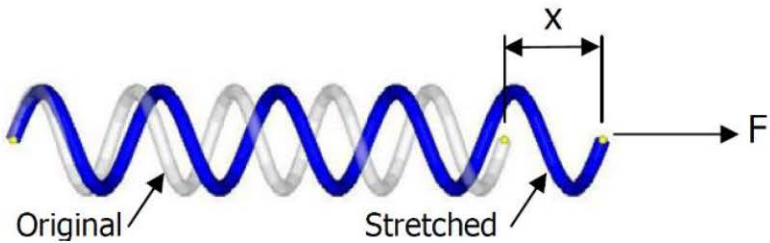


Fig. 112 Spring Equation F=kx [10]

In FEA most structures can be considered as a big, complex spring. However, instead of having to calculate **x** once, like in the simple spring example, the displacement needs to be calculated for eachnode in the model.

The solver is solving literally thousands of these **F=kx** equations simultaneously using the forces and the material stiffness you've defined, to come up with the displacement of every node in the model.

Once it knows the nodal displacements and how each element is deforming, it can also calculate stress within the element. This is particularly handy, as this is what determines whether your part is going to break or not [10].

4.1. General considerations about solving phase[2]

While the pre-processing and post-processing phases of the finite element method are interactive and time-consuming for the analyst, the solution is often a batch process, and is demanding of computer resource.

The governing equations are assembled into matrix form and are solved numerically. The assembly process depends not only on the type of analysis (e.g. static or dynamic), but also on the model's element types and properties, material properties and boundary conditions [47].

In the case of a linear static structural analysis, the assembled equation is of the form:

$$\mathbf{Kd} = \mathbf{r}, \qquad \text{Eq. 15}$$

where **K** is the system stiffness matrix, **d** is the nodal *degree of freedom* (dof) displacement vector, and **r** is the applied nodal load vector. To appreciate this equation, one must begin with the underlying elasticity theory. The strain-displacement relation may be introduced into the stress-strain relation to express stress in terms of displacement.

Under the assumption of compatibility, the differential equations of equilibrium in concert with the boundary conditions then determine a unique displacement field solution, which in turn determines the strain and stress fields. The chances of directly solving these equations are slim to none for anything but the most trivial geometries, hence the need for approximate numerical techniques presents itself.

A finite element mesh is actually a displacement-nodal displacement relation, which, through the element interpolation scheme, determines the displacement anywhere in an element given the values of its nodal dof. Introducing this relation into the strain-displacement relation, we may express strain in terms of the nodal displacement, element interpolation scheme and differential operator matrix.

Recalling that the expression for the potential energy of an elastic body includes an integral for strain energy stored (dependent upon the strain field) and integrals for work done by external forces (dependent upon the displacement field), we can therefore express system potential energy in terms of nodal displacement [47].

---

[2] *The content of this chapter (marked with [47]) was taken from the paper: Roensch, S. "The Finite Element Method:A Four-Article Series" with the written consent of the author, whom I thank.*

Applying the principle of minimum *potential energy*[3], we may set the partial derivative of potential energy with respect to the nodal dof vector to zero, resulting in: a summation of element stiffness integrals, multiplied by the nodal displacement vector, equals a summation of load integrals.

Each stiffness integral results in an element stiffness matrix, which sum to produce the system stiffness matrix, and the summation of load integrals yields the applied load vector, resulting in **Kd = r**. In practice, integration rules are applied to elements, loads appear in the r vector, and nodal dof boundary conditions may appear in the **d** vector or may be partitioned out of the equation [47].

Solution methods for finite element matrix equations are plentiful. In the case of the **linear static Kd = r,** inverting **K** is computationally expensive and numerically unstable. A better technique is Cholesky factorization, a form of Gauss elimination, and a minor variation on the "LDU" factorization theme. The **K** matrix may be efficiently factored into LDU, where L is lower triangular, **D** is diagonal, and **U** is upper triangular, resulting in LDU **d = r**. Since **L** and **D** are easily inverted, and **U** is upper triangular, **d** may be determined by back-substitution.

Another popular approach is the wavefront method, which assembles and reduces the equations at the same time. Some of the best modern solution methods employ sparse matrix techniques. Because node-to-node stiffnesses are non-zero only for nearby node pairs, the stiffness matrix has a large number of zero entries. This can be exploited to reduce solution time and storage by a factor of 10 or more. Improved solution methods are continually being developed. The key point is that the analyst must understand the solution technique being applied [47].

Dynamic analysis for too many analysts means *normal modes*. Knowledge of the natural frequencies and mode shapes of a design may be enough in the case of a single-frequency vibration of an existing product or prototype, with FEA being used to investigate the effects of mass, stiffness and damping modifications. When investigating a future product, or an existing design with multiple modes excited, forced response modeling should be used to apply the expected transient or frequency environment to estimate the displacement and even dynamic stress at each time step [47].

This discussion has assumed h-code elements, for which the order of the interpolation polynomials is fixed. Another technique, p-code, increases the order iteratively until convergence, with error estimates available after one analysis. Finally, the boundary element method places elements only along the geometrical boundary. These techniques have limitations, but expect to see more of them in the near future [47].

---

[3] *See* *Annex A.3*

4.2. Modern design problem

- determining the behavior of a system under the effect of external actions (Fig. 113);
- which is the response (Fig. 114) of the system when subjected to external actions (changes in the forces, temperatures and so on).

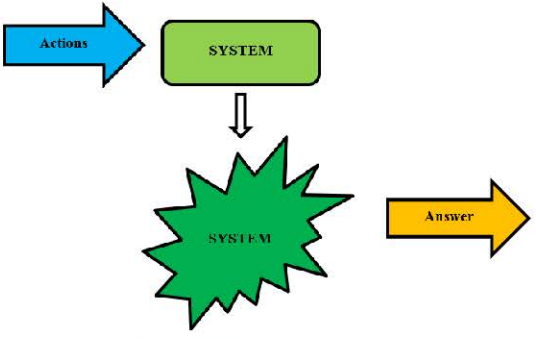


Fig. 113 System reaction

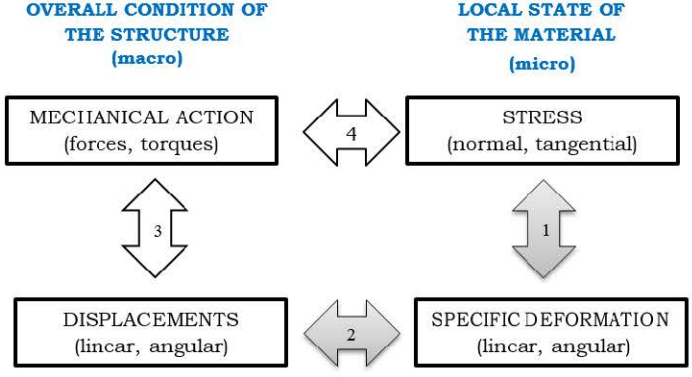


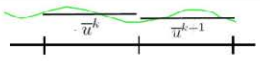
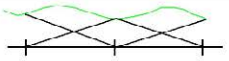
Fig. 114 Internal and external behavior of a system subject to a mechanical stress [44]

4.3. Solving methods

4.3.1. Numerical Methods [56]

1. Finite Element Method

2. Boundary Element Method
3. Finite Difference Method
4. Finite Volume Method
5. Meshless Method

Type	**Finite difference methods**	**Finite volume methods**	**Finite element methods**
Grafical representation			
Comments	Explicit in time Strong theory	The local approximation is a cell average. $\int_{x^{k-1/2}}^{x^{k+1/2}} u_h(x)dx = h^k \bar{u}^k$	The solution is defined in a nonlocal manner. $u_h(x) = \sum_{k=1}^{N} u_k \varphi_k(x)$
Main benefits	Simple to implement and fast	Robust and fast due to locality Complex geometries Well suited for conservation laws Explicit in time	Higher-order accuracy and complex geometries can be combined
Main problem	Simple local approximation and geometric flexibility are not agreeable	Inability to archive high-order accuracy on general grids.	Implicit in time Not well suited for problems with direction.
Complex geometry	X	√	√
Higher-order accuracy and *hp*-adaptivity	√	X	√
Local mass Conservation	√	√	X

4.3.2. Fundamental concepts [55]

Many engineering phenomena can be expressed by "*governing equations*" and "*boundary conditions*".

Elastic problems

Thermal problems

Fluid flow

Electrostatics

Etc.

Governing Equations (Differential equations)

$L(\Phi) + f = 0$

Boundary Conditions

$B(\Phi) + g = 0$

Example: Vertical machining center

- **Elastic deformation**
- **Thermal behaviour**
- **Etc.**

Geometry is very complex!

Governing Equations $L(\Phi) + f = 0$

FEM

A set of simultaneous algebraic equations.

Boundary Conditions $B(\Phi) + g = 0$

Aproximate!

$$[K]\{u\} = \{F\}$$

You know all the equations, but you cannot solve it by hand

$$[K]\{u\} = \{F\} \Rightarrow \{u\} = [K]^{-1}\{F\}$$

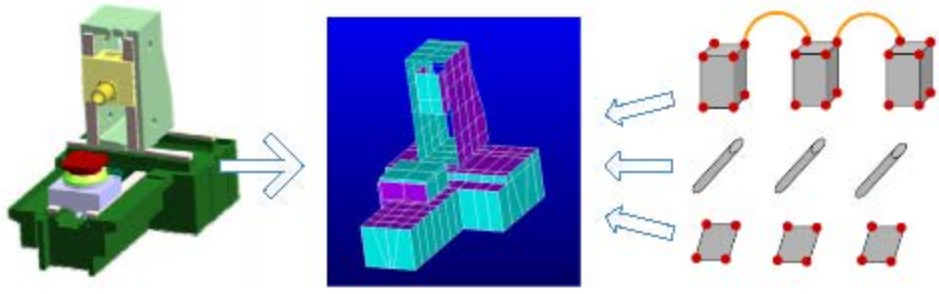
Property Behaviour Action

	Property [K]	Behavior {u}	Action {F}
Elastic	stiffness	displacements	force
Thermal	conductivity	temperature	heat source
Fluid	viscosity	velocity	body force
Electrostatic	dielectric permittivity	electric potential	charge

unknown

It is very difficult to make the algebraic equations for the entire domain

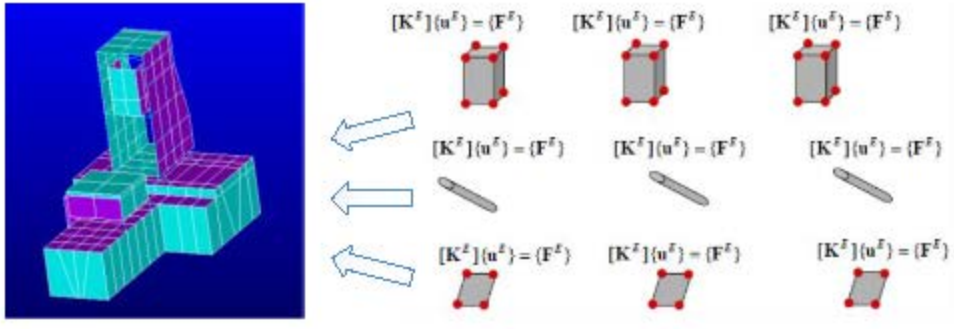
- Divide the domain into a number of small, simple elements
- A field quantity is interpolated by a polynomial over an element
- Adjacent elements share the DOF at connecting nodes



Finite element = small piece of structure

Obtain the algebraic equations for each element (this is easy!).

- Put all the element equations together



Solve the equations, obtaining unknown variabless at nodes.

$[K]\{u\} = \{F\}$ $\Rightarrow$ $\{u\} = [K]^{-1}\{F\}$

Concepts – Summary

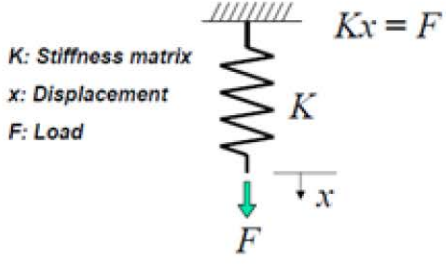
FEM uses the concept of piecewise polynomial interpolation.

By connecting elements together, the field quantity becomes interpolated over the entire structure in piecewise fashion.

A set of simultaneous algebraic equations at nodes.

$$[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\}$$

Property — Behavior — Action



Advantages of the FEM

- ✓ Can readily handle very complex geometry:
  - The heart and power of the FEM
- ✓ Can handle a wide variety of engineering problems
  - Solid mechanics - Dynamics - Heat problems
  - Fluids - Electrostatic problems
- ✓ Can handle complex restraints
  - Indeterminate structures can be solved.
- ✓ Can handle complex loading
  - Nodal load (point loads)
  - Element load (pressure, thermal, inertial forces)
  - Time or frequency dependent loading

Disadvantages of the FEM

- × A general closed-form solution, which would permit one to examine system response to changes in various parameters, is not produced.
- × The FEM obtains only "approximate" solutions.
- × The FEM has "inherent" errors.
- × Mistakes by users can be fatal.

4.3.3. FEM formulation for a linear differential equation

The Finite Element Method (FEM) is a weighted residual method that uses compactly-supported basis functions.

Brief Comparison with Other Methods

Finite Difference Method (FDM):	Finite Element Method (FEM):
FDM approximates an operator (e.g., the derivative) and solves a problem on a set of points (the grid)	FEM uses exact operators but approximates the solution basis functions. Also, FE solves a problem on the interiors of grid cells (and optionally on the gridpoints as well).

Spectral Methods (SM):	Finite Element Method (FEM):
SM use global basis functions to approximate a solution across the entire domain	FEM methods use compact basis functions to approximate a solution on individual elements.

Overview of the Finite Element Method

$$(S) \Leftrightarrow (W) \approx (G) \Leftrightarrow (M)$$

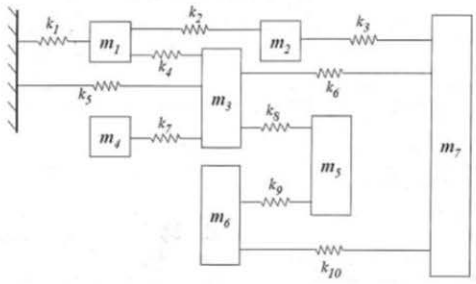
Strong form	Weak form	Galerkin approx	Matrix form

4.3.4. From Strong Form to Weak form (1D)[4] [13]

Classification of Engineering Systems

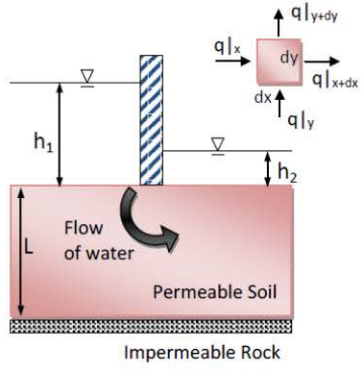
Discrete

[4] *The content of this chapter (marked with [13]) was taken from the paper Chatzi, E.: "The Finite Element Method for the Analysis of Non-Linear and Dynamic Systems", lecture notes, with the written consent of the author, whom I thank.*



Direct Stiffness method: **F = k x**

Continous



Laplace Equation: $$k\left(\frac{\partial^2\phi}{\partial^2 x}+\frac{\partial^2\phi}{\partial^2 y}\right)=0$$

The analysis of characteristic phenomena of continuous environments can be mathematically described in two ways:

Differentially – the solution to the problem is obtained by solving the system of differential equations with respect to initial conditions and / or limits.

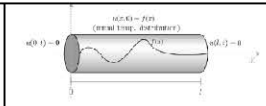
Variationally – the solution to the problem is obtained by seeking a stationary features to make a functional (leading to maximum or minimum) subject to the conditions initial and / or boundary. This method requires knowledge of specific features of the physical phenomenon of the problem.

The functional determination can be made:

- Based on differential equations modeling this phenomenon;
- Based on energy theorems governing the phenomenon.

Consider the following 1 Dimensional (1D) strong form (parabolic)

$$\frac{d}{dx}(c(x)\frac{du}{dx}) + f(x) = 0$$
$$-c(0)\frac{d}{dx}u(0) = C_1$$
$$u(L) = 0$$

Physical problem (1D)		**Diff. Eq.**	**Quantities**	**Constitutive law**
One dimensional Heat flow		$\frac{d}{dx}\left(Ak\frac{dT}{dx}\right) + Q = 0$	T=temperature A=area k=thermal conductivity Q=heat supply	Fourier q = -*k* dT/dx q = heat flux
Axially loaded bar		$\frac{d}{dx}\left(AE\frac{du}{dx}\right) + b = 0$	u=displacement A=area E=Young's Modulus B=axial loading	Hooke $\sigma = E$ du/dx σ = stress

The strong form requires strong continuity on the dependent field variables (usually displacements). Whatever functions define these variables have to be differentiable up to the order of the PDE that exist in the strong form of the system equations. Obtaining the exact solution for a strong form of the system equation is a quite difficult task for practical engineering problems.

The finite difference method can be used to solve the system equations of the string form and obtain an approximate solution. However, this method usually works well for problems with simple and regular geometry and boundary conditions.

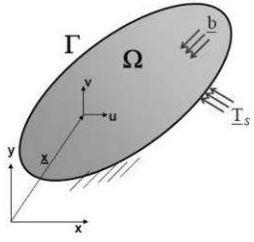
Alternatively we can use the finite element method on a weak form of the system. This is usually obtained through energy principles which is why it is also known as variational form.

Three are the approaches commonly used to go from strong to weak form:

- Principle of Virtual Work;
- Principle of Minimum Potential Energy;
- Methods of weighted residuals (Galerkin, Collocation, Least Squares methods, etc).

4.3.4.1. Approach #1. Principle of Virtual Work

For any set of compatible small virtual displacements imposed on the body in its state of equilibrium, the total internal virtual work is equal to the total external virtual work.



$$W_{int} = \int_{\Omega} \bar{\epsilon}^T \tau d\Omega = W_{ext} = \int_{\Omega} \bar{\mathbf{u}}^T \mathbf{b} d\Omega + \int_{\Gamma} \bar{\mathbf{u}}^{ST} \mathbf{T_S} d\Gamma + \sum_i \bar{\mathbf{u}}^{iT} \mathbf{R_C}^i$$

where

- T_S: surface traction (along boundary Γ)
- b: body force per unit area
- R_C: nodal loads
- $\bar{\mathbf{u}}$: virtual displacement
- $\bar{\epsilon}$: virtual strain
- τ: stresses

4.3.4.2. Approach #2. Principle of Minimum Potential Energy

Applies to elastic problems where since the elasticity matrix is positive definite, hence the energy functional Π has a minimum (stable equilibrium).

Approach #1 applies in general.

The potential energy Π is defined as the strain energy **U** minus the work of the external loads **W**:

$$\mathbf{\Pi} = \mathbf{U} - \mathbf{W}$$

$$\mathbf{U} = \frac{1}{2} \int_{\Omega} \epsilon^T C \epsilon d\Omega$$

$$\mathbf{W} = \int_{\Omega} \bar{\mathbf{u}}^T \mathbf{b} d\Omega + \int_{\Gamma_T} \bar{\mathbf{u}}^{ST} \mathbf{T_s} d\Gamma_T + \sum_i \bar{\mathbf{u}}_i^T \mathbf{R_C}^i$$

(b, T_s, R_C as defined previously)

4.3.4.3. Approach #3. Methods of weighted residuals

Given an arbitrary weight function **w**, where

$$S = \{u | u \in C^0, u(l) = 0\}, S^0 = \{w | w \in C^0, w(l) = 0\}$$

$\mathbf{C^0}$ is the collection of all continuous functions.

Multiplying by w and integrating over Ω

$$\int_0^l w(x)[(c(x)u'(x))' + f(x)]dx = 0$$
$$[w(0)(c(0)u'(0) + C_1] = 0$$

Using the divergence theorem (integration by parts) we reduce the order of the differential:

$$\int_0^l wg'dx = [wg]_0^l - \int_0^l gw'dx$$

The weak form is then reduced to the following problem.

Find $u(x) \in S$ such that:

$$\int_0^l w'cu'dx = \int_0^l wfdx + w(0)C_1$$
$$S = \{u | u \in C^0, u(l) = 0\}$$
$$S^0 = \{w | w \in C^0, w(l) = 0\}$$

Stages:

1. Analytical modeling
2. Geometric domain modeling

3. Physical parameters modeling (approximation)
4. Numerical modeling using finite elements
5. Numerical model solving
6. Program compiling and result analysis

FEM involves, instead of directly solving differential equations, determining the approximate solution of an equivalent integral form. The procedures used for the deduction of the numerical model finite element can be grouped as follows:

- ✓ Direct method ;
- ✓ Variational method;
- ✓ Weighted residue method;
- ✓ Energetic method.

4.4. Types of analysis

- Structural
- Thermal
- Electromagnetic

The differential equations to describe the structure behavior of a infinitesimal particle

- for resistance problems > Theory of elasticity
- for fluid mechanics > Navier-Stokes equations
- for magnetic fields > Maxwell equations
- for heat transfer in solids > Fourier equation

The function described by the differential equation is a particular measurement:

- for resistance problems > displacement
- for fluid mechanics > speed, pressure

- for magnetic fields > magnetic potential
- for heat transfer > temperature

In order to solve by the classical approach, after rough approximations on geometry, the initial and boundary conditions and material properties applied to theoretical mathematical and calculation models, it results simplified analytical mathematical models which can be processed using manual calculation, slide ruler or a pocket calculator.

For instance, the calculation model of mechanical structures elastically deformable, using theory of elasticity and strength of materials methods, specific mathematical models are obtained and lead to simple calculating relationships (Navier, Juravski and so on) for different geometrical areas (bars, boards, membranes, tubes, discs, etc..) and specific physical conditions.

In order to increasing the precision of the results obtained by classical methods, the numerical methods through small approximations, usually controllable, in terms of geometry, the boundary conditions and material properties lead to numerical models that can be solved only by a numerical computer.

4.4.1. Linear Static Analysis

Most structural analysis problems can be treated as linear static problems, based on the following assumptions:

1. Small deformations (loading pattern is not changed due to the deformed shape)

2. Elastic materials (no plasticity or failures) - Fig. 115, between O and A.

3. Static loads (the load is applied to the structure in a slow or steady fashion)

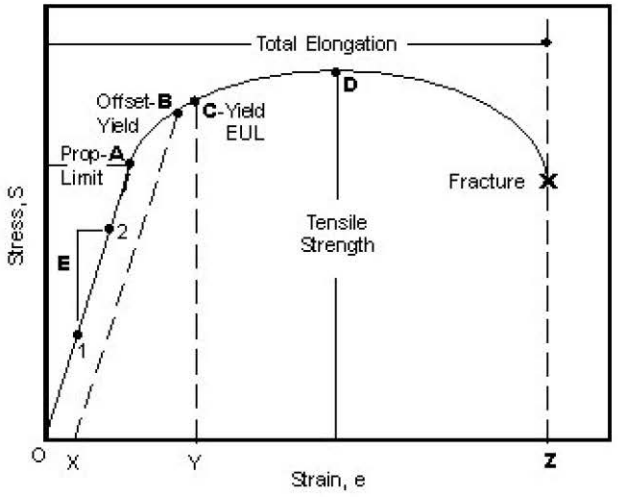


Fig. 115 Stress – strain curve

Linear analysis can provide most of the information about the behavior of a structure, and can be a good approximation for many analyses. It is also the bases of nonlinear analysis in most of the cases.

4.4.1.1. Stress

In continuum mechanics, stress is a physical quantity that expresses the internal forces that neighboring particles of a continuous material exert on each other. For example, when a solid vertical bar is supporting a weight, each particle in the bar pulls on the particles immediately above and below it. When a liquid is under pressure, each particle gets pushed inwards by all the surrounding particles, and, in reaction, pushes them outwards. These macroscopic forces are actually the average of a very large number of intermolecular forces and collisions between the particles in those molecules.

Stress inside a body may arise by various mechanisms, such as reaction to external forces applied to the bulk material (like gravity) or to its surface (like contact forces, external pressure, or friction). Any strain (deformation) of a solid material generates an internal elastic stress, analogous to the reaction force of a spring, that tends to restore the material to its original undeformed state. In liquids and gases, only deformations that change the volume generate persistent elastic stress. However, if the deformation is gradually changing with time, even in fluids there will usually be some viscous stress, opposing that change. Elastic and viscous stresses are usually combined under the name mechanical stress [25].

Significant stress may exist even when deformation is negligible or non-existent (a common assumption when modeling the flow of water). Stress may exist in the absence of external forces; such built-in stress is important, for example, in prestressed concrete and tempered glass. Stress may also be imposed on a material without the application of net forces, for example by changes in temperature or chemical composition, or by external electromagnetic fields (as in piezoelectric and magnetostrictive materials).

The relation between mechanical stress, deformation, and the rate of change of deformation can be quite complicated, although a linear approximation may be adequate in practice if the quantities are small enough. Stress that exceeds certain strength limits of the material will result in permanent deformation (such as plastic flow, fracture, cavitation) or even change its crystal structure and chemical composition.

In some branches of engineering, the term stress is occasionally used in a looser sense as a synonym of "internal force". For example, in the analysis of trusses, it may refer to the total traction or compression force acting on a beam, rather than the force divided by the area of its cross-section [25].

The stress is the force acting on the surface of a body, the surface tends to zero. Stresses are classified according to the direction of force on the body surface (**Fig. 116**):

- normal stress, which is denoted by σ, arises from the force vector component perpendicular or antiparallel to the material cross section on which it acts.
- a shear stress which is denoted as τ, is defined as the component of stress coplanar with a material cross section. Shear stress arises from the force vector component parallel to the cross section.

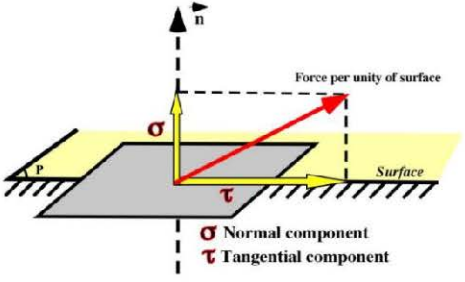


Fig. 116 Stress components [25]

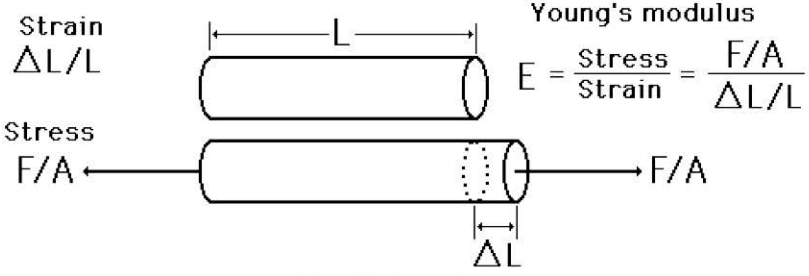


Fig. 117 Elongation

The formula to calculate average of the general shear stress is:

$$\tau = \frac{F}{A},$$

Eq. 16

where

τ = the shear stress;

F = the force applied;

A = the cross-sectional area of material with area parallel to the applied force vector.

4.4.1.2. Strain

Under a tension body deforms. Deformation can change the volume or body shape. Also, the elastic deformation can be recovered in the moment of unloading the body, or can be the flow and remains unrecovered.

There are various kinds of deformations:

- compression - the normal stresses are oriented inside the body and volume decreases
- dilation - the normal stresses are oriented outside the body and volume increases
- simple shear
- pure shear
- rotation

Compression and expansion are part of volume deformations.

Strain is defined as "deformation of a solid due to stress" and can be expressed as (Fig. 117):

$$\varepsilon = \Delta l / l = \sigma / E, \qquad \textbf{Eq. 17}$$

where

ε = unitless measure of engineering strain;

Δl = length variation;

l = initial length.

E = Young's modulus (Modulus of Elasticity) (N/m^2, Pa). Young's modulus can be used to predict the elongation or compression of an object.

Young's Modulus

Most metals have deformations that are proportional with the imposed loads over a range of loads. Stress is proportional to load and strain is proportional to deformation expressed by the Hooke's law like

$$E = \text{stress} / \text{strain} = (F_n / A) / (dl / l_o). \qquad \textbf{Eq. 18}$$

Modulus of Elasticity or Young's Modulus are commonly used for metals and metal alloys and expressed in terms 10^6 N/m^2 or Pa. Tensile modulus are often used for plastics and expressed in terms 10^5 N/m^2 or Pa.

Shear Modulus

$$S = \text{stress} / \text{strain} = (F_p / A) / (S / d) \qquad \text{Eq. 19}$$

where

S = shear modulus (N/m^2)
F_p = force parallel to the faces which they act
A = area (m^2)
s = displacement of the faces (m)
d = distance between the faces displaced (m)

4.4.1.3. Stress analysis

Stress analysis is a branch of applied physics that covers the determination of the internal distribution of stresses in solid objects. It is an essential tool in engineering for the study and design of structures such as tunnels, dams, mechanical parts, and structural frames, under prescribed or expected loads. It is also important in many other disciplines; for example, in geology, to study phenomena like plate tectonics, vulcanism and avalanches; and in biology, to understand the anatomy of living beings [25].

Stress analysis is generally concerned with objects and structures that can be assumed to be in macroscopic static equilibrium. By Newton's laws of motion, any external forces are being applied to such a system must be balanced by internal reaction forces, which are almost always surface contact forces between adjacent particles — that is, as stress. Since every particle needs to be in equilibrium, this reaction stress will generally propagate from particle, creating a stress distribution throughout the body.

The typical problem in stress analysis is to determine these internal stresses, given the external forces that are acting on the system. The latter may be body forces (such as gravity or magnetic attraction), that act throughout the volume of a material; or concentrated loads (such as friction between an axle and a bearing, or the weight of a train wheel on a rail), that are imagined to act over a two-dimensional area, or along a line, or at single point.

In stress analysis one normally disregards the physical causes of the forces or the precise nature of the materials. Instead, one assumes that the stresses are related to deformation (and, in non-static problems, to the rate of deformation) of the material by known constitutive equations [25].

Stress analysis may be carried out experimentally, by applying loads to the actual artifact or to scale model, and measuring the resulting stresses, by any of several available methods. This approach is often used for safety certification and monitoring. However, most stress analysis is done by mathematical methods, especially during design.

The basic stress analysis problem can be formulated by Euler's equations of motion for continuous bodies (which are consequences of Newton's laws for conservation of linear momentum and angular momentum) and the Euler-Cauchy stress principle, together with the appropriate constitutive equations. Thus one obtains a system of partial differential equations involving the stress tensor field and the strain tensor field, as unknown functions to be determined. The external body forces appear as the independent ("right-hand side") term in the differential equations, while the concentrated forces appear as boundary conditions. The basic stress analysis problem is therefore a boundary-value problem.

Stress analysis for elastic structures is based on the theory of elasticity and infinitesimal strain theory. When the applied loads cause permanent deformation, one must use more complicated constitutive equations, that can account for the physical processes involved (plastic flow, fracture, phase change, etc.).

However, engineered structures are usually designed so that the maximum expected stresses are well within the range of linear elasticity (the generalization of Hooke's law for continuous media); that is, the deformations caused by internal stresses are linearly related to them. In this case the differential equations that define the stress tensor are linear, and the problem becomes much easier. For one thing, the stress at any point will be a linear function of the loads, too. For small enough stresses, even non-linear systems can usually be assumed to be linear.

Stress analysis is simplified when the physical dimensions and the distribution of loads allow the structure to be treated as one- or two-dimensional (Fig. 118). In the analysis of trusses, for example, the stress field may be assumed to be uniform and uniaxial over each member. Then the differential equations reduce to a finite set of equations (usually linear) with finitely many unknowns. In other contexts one may be able to reduce the three-dimensional problem to a two-dimensional one, and/or replace the general stress and strain tensors by simpler models like uniaxial tension/compression, simple shear, etc.

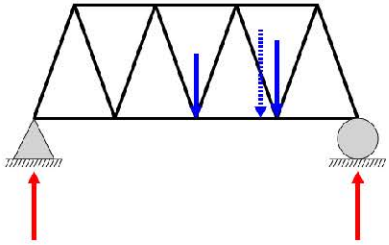


Fig. 118 Simplified model of a truss for stress analysis,

Still, for two- or three-dimensional cases one must solve a partial differential equation problem. Anlytical or closed-form solutions to the differential equations can be obtained when the geometry, constitutive relations, and boundary conditions are simple enough. Otherwise one must generally resort to numerical approximations such as the finite element method, the finite difference method, and the boundary element method [25].

4.4.2. Vibration Analysis [1]

4.4.2.1. Introduction

A spring and a mass interact with one another to form a system that resonates at their characteristic natural frequency. If energy is applied to a spring-mass system, it will vibrate at its natural frequency. The level of a general vibration depends on the strength of the energy source as well as the damping inherent in the system. Consider the single degree of freedom system in Fig. 119 that is usually introduced in a first course in physics or ordinary differential equations.

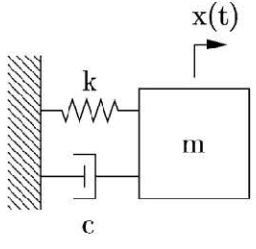


Fig. 119 A spring-mass-damper single degree of freedom system

There, k is the spring constant, or stiffness, and m is the mass, and c is a viscous damper. If the system is subjected to a horizontal force, say ***f(t)***, then Newton's law of motion leads to the differential equation of motion in terms of the displacement as a function of time, **x(t)**:

$$m\, d^2x/dt^2 + c\, dx/dt + k\, x(t) = f(t) \qquad \textbf{Eq. 20}$$

which requires the initial conditions on the displacement, ***x(0)***, and velocity, ***v(0) = dx / dt(0)***. When there is no external force and no damping, then it is called free, undamped motion, or simple harmonic motion (SHM):

$$m\, d^2x/dt^2 + k\, x(t) = 0. \qquad \textbf{Eq. 21}$$

The usual simple harmonic motion assumption is $x(t) = a \sin(\omega t)$ where a is the amplitude of motion and ω is the circular frequency of the motion. Then the motion is described by

$$[k - \omega^2 m]\, a \sin(\omega t) = 0, \text{ or } [k - \omega^2 m] = 0. \qquad \textbf{Eq. 22}$$

The above equation represents the simplest eigen-analysis problem. There you wish to solve for the eigenvalue, **ω**, and the eigenvector, ***a***. Note that the amplitude, a, of the eigenvector is not known. It is common to scale the eigenvector to make the largest amplitude unity. The above scalar problem is easily solved for the circular frequency (eigenvalue),

$$\omega = 2\pi F_n = \sqrt{k/m}, \qquad \textbf{Eq. 23}$$

which is related to the so called natural frequency, $\mathbf{F_n}$, by $\mathbf{F_n = \omega / 2\pi}$.

From this, it is seen that if the stiffness increases, the natural frequency also increases, and if the mass increases, the natural frequency decreases. If the system has damping, which all physical systems do, its frequency of response is a little lower, and depends on the amount of damping. They can be useful in validating finite element calculations. Note that the above simplification neglected the mass of both the spring and the dampener. Any physical structure vibration can be modeled by springs (stiffnesses), masses, and dampers. In elementary models you use line springs and dampers, and point masses. It is typical to refer to such a system as a "lumped mass system". For a continuous part, both its stiffness

and mass are associated with the same volume. In other words, a given volume is going to have a strain energy associated with its stiffness and a kinetic energy associated with its mass. A continuous part has mass and stiffness matrices that are of the same size (have the same number of DOF). The mass contributions therefore interact and can not naturally be lumped to a single value at a point. There are numerical algorithms to accomplish such a lumped (or diagonal) mass matrix but it does not arise in the consistent finite element formulation

4.4.2.2. Finite Element Vibration Studies

In finite element models, the continuous nature of the stiffness and mass leads to the use of square matrices for stiffness, mass, and damping. They can still contain special cases of line element springs and dampers, as well as point masses. Dampers dissipate energy, but springs and masses do not.

If you have a finite element system with many DOF then the above single DOF system generalizes to a displacement vector, **X(t)** interacting with a square mass matrix, **M**, stiffness matrix, **K**, damping matrix **C**, and externally applied force vector, F(t), but retains the same general form:

$$M\,d^2X/dt^2 + C\,dX/dt + K\,X(t) = F(t) \qquad \text{Eq. 24}$$

plus the initial conditions on the displacement, **X(0)**, and velocity, **v(0) = dX / dt(0)**. Integrating these equations in time gives a time history solution. The solution concepts are basically the same, they just have to be done using matrix algebra. The corresponding SHM, or free vibration mode (**C = 0**, **F = 0**) for a finite element system is

$$M\,d^2X/dt^2 + K\,X(t) = 0. \qquad \text{Eq. 25}$$

The SHM assumption generalizes to $X(t) = A \sin(\omega t)$ where the amplitude, **A**, is usually called the mode shape vector at circular frequency **ω**. This leads to the general matrix eigenvalue problem of a zero determinant:

$$|K - \omega^2 M| = 0. \qquad \text{Eq. 26}$$

There is a frequency, say ω_k, and mode shape vector, A_k, for each degree of freedom, **k**. A matrix eigenvalue-eigenvector solution is much more computationally expensive that a matrix time history solution. Therefore most finite element systems usually solve for the first few natural

frequencies. Depending on the available computer power, that may mean 10 to 100 frequencies. A lot of FEM software applications includes natural frequency and mode shape calculations as well as time history solutions.

Usually you are interested only in the first few natural frequencies. A zero natural (or slightly negative one) frequency corresponds to a rigid body motion. A part or assembly has at most six RBM of 'vibration', depending on how or if it is supported. If a shell model is used the rotational DOF exist and the mass matrix is generalized to include the mass moments of inertia. For every natural frequency there is a corresponding vibration mode shape. Most mode shapes can generally be described as being an axial mode, torsional mode, bending mode, or general mode.

Like stress analysis models, probably the most challenging part of getting accurate finite element natural frequencies and mode shapes is to get the type and locations of the restraints correct. A crude mesh will give accurate frequency values, but not accurate stress values. The solver software contains equations for most known analytic solutions for the frequencies of mechanical systems. They can be quite useful in validating the finite element frequency results.

4.4.2.3. Analytic Solutions for Frequencies

The analytic frequency and mode shape solutions for many parts with common geometries are found In a course on the vibration of continuous media. The geometries include axial bars, axial shafts in torsion, beams with transverse motion vibration, flat plates of various shapes, and thin shells of various shapes. Several examples of them are given in the "validation problems" set of examples presented along side the software tutorials.

Consider the longitudinal vibration of a bar. The results depend on which type of support is applied to each end of the bar. For one end restrained and the other end free the natural frequencies are

$$\omega_n = \frac{(2n-1)\pi c}{2L},\ c = \sqrt{\frac{E}{\rho}},\ n = 1, 2, 3, \ldots \infty. \qquad \text{Eq. 27}$$

However, if both ends are restrained they are

$$\omega_n = \frac{n\pi c}{L},\ c = \sqrt{\frac{E}{\rho}},\ n = 1, 2, 3, \ldots \infty.$$ **Eq. 28**

This shows that for a continuous body there are, in theory, an infinite number of natural frequencies and mode shapes. Try a single quadratic element to model a fixed-fixed bar frequency. Restrain the two end DOF (the first and third row and column) of the above 3 by 3 matrices. Only a single DOF remains to approximate the first mode. Solve the restrained matrix eigen-problem:

$$|[k] - \omega^2[m]| = 0.$$ **Eq. 29**

The reduced terms in the matrices are

$$\frac{EA}{3L}[16] - \omega^2\frac{\rho AL}{30}[16] = 0$$

So, $\omega_1^2 = \frac{10E}{L^2\rho}$ and $\omega_1 = \sqrt{10}\frac{c}{L} = 3.16\frac{c}{L}$,

which is less that 1% error compared to the exact result. Adding more elements increases the accuracy of each frequency estimate, and also yields estimates of the frequencies associated with the additional DOF. For example, adding a second quadratic bar element gives a total of three un-restrained DOF. So you could solve for the first three frequencies. The value for ω_1 would be more accurate and you would have the first estimates of ω_2 and ω_3.

Usually, the masses farthest from the supports have the most effects on the natural frequency calculations. If you only care about the frequencies you could use split lines to build larger elements near the supports. For beams and shells, the transverse displacements are more important than the tangential rotational DOF.

4.4.3. Buckling Analysis [1]

4.4.3.1. Introduction

There are two major categories leading to the sudden failure of a mechanical component: material failure and structural instability, which is often called buckling. For material failures you need to consider the yield stress for ductile materials and the ultimate stress for brittle materials.

Those material properties are determined by axial tension tests and axial compression tests of short columns of the material (see Fig. 120). The geometry of such test specimens has been standardized. Thus, geometry is not specifically addressed in defining material properties, such as yield stress. Geometry enters the problem of determining material failure only indirectly as the stresses are calculated by analytic or numerical methods.

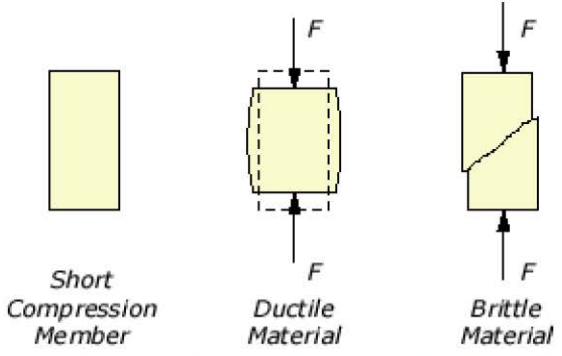


Fig. 120 *Short columns fail due to material failure [1]*

Predicting material failure may be accomplished using linear finite element analysis. That is, by solving a linear algebraic system for the unknown displacements, $\mathbf{K}\cdot\boldsymbol{\delta} = \mathbf{F}$. The strains and corresponding stresses obtained from this analysis are compared to design stress (or strain) allowables everywhere within the component. If the finite element solution indicates regions where these allowables are exceeded, it is assumed that material failure has occurred.

The load at which buckling occurs depends on the stiffness of a component, not upon the strength of its materials. Buckling refers to the loss of stability of a component. The buckling mode is usually independent of material strength. This loss of stability usually occurs within the elastic range of the material.

The two phenomenon are governed by different differential equations. Buckling failure is primarily characterized by a loss of structural stiffness and is not modeled by the usual linear finite element analysis, but by a finite element eigenvalue-eigenvector solution, $|\mathbf{K} + \lambda_m \cdot \mathbf{K_F}|\ \delta_m = 0$, where λ_m is the buckling load factor (BLF) for the **m**-th mode, $\mathbf{K_F}$ is the additional "geometric stiffness" due to the stresses caused by the loading, **F**, and δ_m is the associated buckling displacement shape for the **m**-th mode. The spatial distribution of the load is important, but its relative magnitude is not. The buckling calculation gives a multiplier that scales the magnitude of the load (up or down) to that required to cause buckling. The multiplier depends on the material modulus.

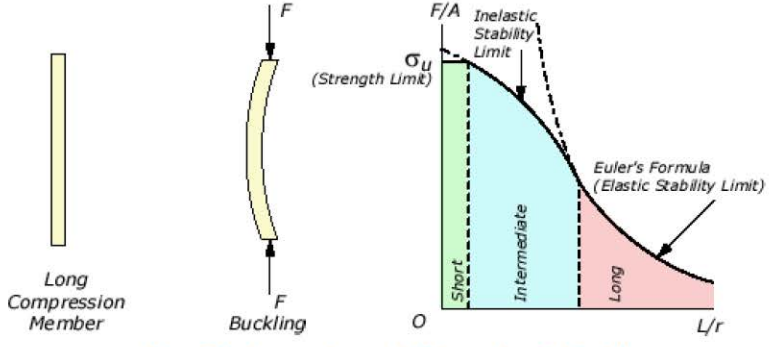


Fig. 121 *Long columns fail due to instability [1]*

Slender or thin-walled components under compressive stress are susceptible to buckling. Most people have observed what is called "Euler buckling" where a long slender member subject to a compressive force moves lateral to the direction of that force, as illustrated in Fig. 121. The force, **F**, necessary to cause such a buckling motion will vary by a factor of four depending only on how the two ends are restrained. Therefore, buckling studies are much more sensitive to the component restraints that in a normal stress analysis. The theoretical Euler solution will lead to infinite forces in very short columns, and that clearly exceeds the material ultimate stress. In practice, Euler column buckling can

only be applied to long columns and empirical transition equations are required for intermediate length columns. For very long columns the loss of stiffness occurs at stresses far below the material ultimate or yield stresses.

There are many analytic solutions for idealized components having elastic instability. About 75 of the most common cases are tabulated in the classic references. Euler long column buckling is quite sensitive to the end restraints. Fig. 122 shows five of several cases of end restraints and the associated k value used in computing buckling load or stress.

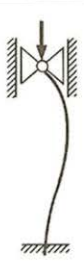
Case	1	2	3	4	5
Constraints					
k	4	1	.25	2.046	1

Fig. 122 *Restraints have a large influence on the critical buckling load [1]*

4.4.3.2. Buckling Terminology

The topic of buckling is still unclear because the keywords of "stiffness", "long" and "slender" have not been quantified. Most of those concepts were developed historically from 1D studies. You need to understand those terms even though finite element analysis lets you conduct buckling studies in 1D, 2D, and 3D. For a material, stiffness refers to either its elastic modulus, **E**, or to its shear modulus, $G = E / (2 + 2\,\nu)$ where ν is Poisson's ratio.

Slender is a geometric concept addressing the ratio of a members length and a property of the cross-sectional area that is quantified by the radius of gyration. The radius of gyration, **r**, has the units of length and describes the way in which the area of a cross-section is distributed

around its centroidal axis. If the area is concentrated far from the centroidal axis it will have a greater value of the radius of gyration and a greater resistance to buckling. A non-circular cross-section will have two values for its radius of gyration. The section tends to buckle around the axis with the smallest value. The radius of gyration, **r**, is defined as:

$$r = \sqrt{I/A}, \qquad \text{Eq. 30}$$

where **I** and **A** are the area moment of inertia, and area of the crosssection. For a circle of radius **R**, you obtain $r = R/2$. Solids can have regions that are slender, and if they carry compressive stresses a buckling study is justified. Long is also a geometric concept that is quantified by the nondimensional "slenderness ratio" **L / r**, where **L** denotes the length of the component. The slenderness ratio, of a part made of a single material, is defined to be long when it is greater than $\pi/k\sqrt{2E/\sigma_y}$, where σ_y is the material yield stress. A long slenderness ratio is typically greater than 120. The above equation is the dividing point between long (Euler) columns and intermediate (empirical) columns. The critical compressive stress that will cause buckling always decreases as the slenderness ratio increases. The critical Euler buckling stress depends on the material, the slenderness ratio, and the end restraint conditions.

4.4.3.3. Buckling Load Factor

The buckling load factor (**BLF**) is an indicator of the factor of safety against buckling or the ratio of the buckling loads to the currently applied loads. Table from Fig. 123 illustrates the interpretation of possible **BLF** values returned by SW Simulation, for example. Since buckling often leads to bad or even catastrophic results, you should utilize a high factor of safety for buckling loads (say **BLF > 2**).

BLF	Buckling Status	Remarks
>1	Buckling not predicted	The applied loads are less than the estimated critical loads.
= 1	Buckling predicted	The applied loads are exactly equal to the critical loads. Buckling is expected.
< 1	Buckling predicted	The applied loads exceed the estimated critical loads. Buckling will occur.
-1 < BLF < 0	Bucklin possible	Buckling is predicted if you reverse the load directions.
-1	Buckling possible	Buckling is expected if you reverse the load directions.
< -1	Buckling not predicted	The applied loads are less than the estimated critical loads, even if you reverse their directions.

Fig. 123 *Interpretation of the Buckling Load Factor*

4.4.3.4. General Buckling Concepts

Other 1D concepts that relate to stiffness are: axial stiffness, **E·A / L**, flexural (bending) stiffness, **E·I / L**, and torsional stiffness, **G·J / L**, where **J** is the polar moment of inertia of the cross-sectional area ($J = I_z = I_x + I_y$). Today, stiffness usually refers to the finite element stiffness matrix, which can include all of the above stiffness terms plus general solid or shell stiffness contributions. Analytic buckling studies identify additional classes of instability besides Euler buckling (see Fig. 124). They include lateral buckling, torsional buckling, and other buckling modes. A finite element buckling study determines the lowest buckling factors and their corresponding displacement modes. The amplitude of a buckling displacement mode, $|\delta_m|$, is arbitrary and not useful, but the shape of the mode can suggest whether lateral, torsional, or other behavior is governing the buckling response of a design.

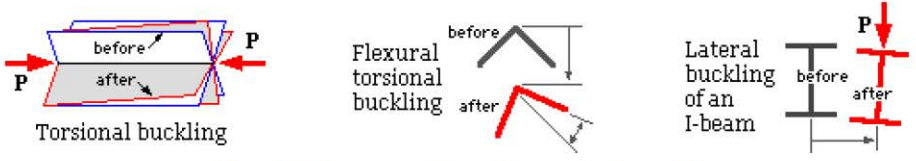


Fig. 124 *Some sample buckling mode shapes [1]*

4.4.4. Thermal Analysis [1]

4.4.4.1. Concepts of thermal analysis

There are three different types of heat transfer: conduction, convection, and radiation. A temperature difference must exist for heat transfer to occur. Heat is always transferred in the direction of decreasing temperature. Temperature is a scalar, but heat flux is a vector quantity. The thermal variables and boundary conditions relate to the displacements and stress in an axial bar through the analogy as summarized in Fig. 125.

Thermal Analysis Item, [units], symbol	Structural Analysis Item, [units], symbol
Unknown: Temperature [K], T	Unknown: Displacements [m], u
Gradient: Temperature Gradient [K/m], ∇T	Gradient: Strains [m/m], ε
Flux: Heat flux [W/m2], q	Flux: Stresses [N/m2], σ
Source: Heat Source for point, line, surface, volume [W], [W/m], [W/m2], [W/m3], Q	Source: Force for point, line, surface, volume [N], [N/m], [N/m2], [N/m3], g
Indirect restraint: Convection	Indirect restraint: Elastic support
Restraint: Prescribed temperature [K], T	Restraint: Prescribed displacement [m], u
Reaction: Heat flow resultant [W], H	Reaction: Force component [N], F
Material Property: Thermal conductivity [W/m-K], k	Material Property: Elastic modulus [N/m2], E
Material Law: Fourier's law	Material Law: Hooke's Law

Fig. 125 *Terms of the 1D thermal-structural analogy [1]*

Conduction takes place within the boundaries of a body by the diffusion of its internal energy. The temperature within the body, **T**, is given in units of degrees Celsius [C], Fahrenheit [F], Kelvin [K], or Rankin [R]. Its variation in space defines the temperature gradient vector, $\nabla \mathbf{T}$, with units of [K/m] say. The heat flux vector, q, per unit area is define by Fourier's Conduction Law, as the thermal conductivity matrix, **k**, times the negative of the temperature gradient, $\mathbf{q} = -\mathbf{k}\,\nabla \mathbf{T}$. The integral of the heat flux over an area yields the total heat flow for that area.

Thermal conductivity has the units of [*W/m-K*] while the heat flux has units of [W/m^2]. The conductivity, **k**, is usually only known to three or four significant figures. For solids it ranges from about 417 *W/m-K* for silver down to 0.76 *W/m-K* for glass. A perfect insulator material ($\boldsymbol{k} = \boldsymbol{0}$) will not conduct heat; therefore the heat flux vector must be parallel to the insulator surface. A plane of symmetry (where the geometry, k values, and heat sources are mirror images) acts as a perfect insulator. In finite element analysis, all surfaces default to perfect insulators unless you give a specified temperature, a known heat influx, a convection condition, or a radiation condition.

Convection occurs in a fluid by mixing. Here we will consider only free convection from the surface of a body to the surrounding fluid. Forced convection, which requires a coupled mass transfer, will not be considered. The magnitude of the heat flux normal to a solid surface by free convection is

$$q_n = h \cdot A_h (T_h - T_f) \qquad \text{Eq. 31}$$

where **h** is the convection coefficient, A_h is the surface area contacting the fluid, T_h is the convecting surface temperature, and T_f is the surrounding fluid temperature, respectively. The units of **h** are $[W/m^2\text{-}K]$. Its value varies widely and is usually known only from one to four significant figures. Typical values for convection to air and water are 5-25 and 500-1000 $W/m^2\text{-}K$, respectively.

Radiation heat transfer occurs by electromagnetic radiation between the surfaces of a body and the surrounding medium. It is a highly nonlinear function of the absolute temperatures of the body and medium. The magnitude of the heat flux normal to a solid surface by radiation is

$$q_r = \varepsilon \cdot \sigma \cdot A_r (T_r^{\,4} - T_m^{\,4}). \qquad \text{Eq. 32}$$

Here T_r is the absolute temperature of the body surface, T_m is the absolute temperature of the surrounding medium, A_r is the body surface area subjected to radiation,

$$\sigma = 5.67 \text{ x } 108 \text{ W/m}^2\text{–K}^4 \qquad \text{Eq. 33}$$

is the Stefan-Boltzmann constant, and ε is a surface factor ($\varepsilon = 1$ for a perfect black body).

Transient, or unsteady, heat transfer in time also requires the material properties of specific heat at constant pressure, c_p in $[kJ/kg\text{-}K]$, and the mass density, ρ in $[kg/m^3]$. The specific heat is typically known to 2 or 3 significant figures, while the mass density is probably the most accurately known material property with 4 to 5 significant figures.

The one-dimensional governing differential equation for transient heat transfer through an area **A**, of conductivity k_x, density ρ, specific heat **cp** with a volumetric rate of heat generation, **Q**, for the temperature **T** at time **t** is

$$\partial (k_x \, \partial T / \partial x) / \partial x + Q(x) = \rho \, c_p \, \partial T / \partial t, \qquad \text{Eq. 34}$$

for $0 \le x \le L$ and time $t \ge 0$.

It requires initial conditions to describe the beginning state, and boundary conditions for later times. For a steady state condition ($\partial T/\partial t = 0$) the typical boundary conditions of one of the following:

1. **T** prescribed at 0 and L, or
2. **T** prescribed at one end and a heat source at the other, or
3. **T** prescribed at one end and a convection condition at the other, or
4. A convection condition at one end and a heat source at the other, or
5. A convection condition at both ends.

In the 3D case the differential equation becomes the anisotropic Poisson Equation. That is, the above diffusion term (second derivatives in space) is expanded to include derivatives with respect to y and z, times their corresponding thermal conductivity values.

4.4.4.2. Finite Element Thermal Analysis

The finite element method creates a set of algebraic equations by using an equivalent governing integral form that is integrated over a mesh that approximates the volume and surface of the body of interest. The mesh consists of elements connected to nodes. In a thermal analysis, there will be one simultaneous equation for each node. The unknown at each node is the temperature. Today, a typical thermal mesh involves 20,000 to 100,000 nodes and thus temperature equations. The restraints are specified temperatures (or a convection condition since it includes a specified fluid temperature). The reactions are is the resultant heat flow that is necessary to maintain a specified temperature. All other conditions add load or source terms. The default surface condition is an insulated boundary, which results in a zero source (load) term.

The assembled matrix equations for thermal equilibrium will be [1]:

$$\begin{bmatrix} K_{uu} & K_{ug} \\ K_{gu} & K_{gg} \end{bmatrix} \begin{Bmatrix} T_u \\ T_g \end{Bmatrix} = \begin{Bmatrix} F_g \\ F_u \end{Bmatrix} \qquad \text{Eq. 35}$$

where now T_g represents the given (restrained) nodal temperatures, F_g represents the known resultant nodal heat power (heat flow) at the node. This system of equations is solved for unknowns T_u. The thermal reactions, F_u, at the given temperature nodes represent the total heat flow, in

or out, necessary to maintain the given temperatures, T_g. From the above structural-thermal analogy, the matrix equations of a linear (temperature interpolation) conducting element is

$$k \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \qquad \text{Eq. 36}$$

where

$$k \equiv k_x A/L \qquad \text{Eq. 37}$$

may be referred to as the thermal stiffness of the rod of length, L, area , A, and thermal conductivity k_x. In this case, **T** corresponds to a nodal temperature, and **F** corresponds to the resultant nodal heat power from the various heat sources. The thermal load (source) items for steady state analysis are given in Fig. 126. Both convection and radiation require inputs of the estimated surface conditions.

Load Type	Geometry	Required Input
Convection	Faces	Film coefficient and bulk temperature
Heat Flux	Faces	Heat flux (heat power/unit area) value
Heat Power	Pts, edges, faces, parts	Total heat power value (rate of heat generation per unit volume times the part volume)
Insulated (Adiabatic)	Faces	None. This is the *default condition* for any face not subject to one of the three above conditions
Radiation	Faces	Surrounding temperature, emissivity values, and view factor for surface to ambient radiation

Fig. 126 Loads for steady state thermal analysis *[1]*

Table from Fig. 127 gives typical convection coefficients values. Note that there is a wide range in such data. Therefore, you will often find it necessary to run more than one study to determine the range of answers that can be developed in your thermal study.

Fluid Medium	h
Air (natural convection)	5-25
Air / superheated steam (forced convection)	10-500
Oil (forced convection)	60-1800
Steam (condensing)	5000-120,000
Water (boiling)	2500-60,000
Water (forced convection)	300-6000

Fig. 127 Typical heat convection coefficient values, h, *[W/m² K] [1]*

Having supplied all the restraints, loads, and properties you can run a thermal analysis and continue on to post-processing and documenting the results. Table from Fig. 128 gives the thermal restraints items for steady state analysis. Most programs offer only a temperature restraint. Different types of FEA software, such as ANSYS or SW, also offers the ability to define a non-ideal material interface, as in Fig. 129. That is often needed in practice and is referred to as a contact resistance. It basically defines a temperature jump across an interface for a given heat flux through the interface. The necessary resistance input, **R**, depends on various factors. Table from Fig. 130 gives typical **R** values, while Table from Fig. 131 cites values of its reciprocal, the conductance.

Restraint	Geometric Entities	Required Input
Temperature	Vertexes, edges, faces and parts	Temperature value
Contact resistance	Two contacting faces. See discussion.	Total or unit thermal resistance.

Fig. 128 Restraints in steady state thermal analysis

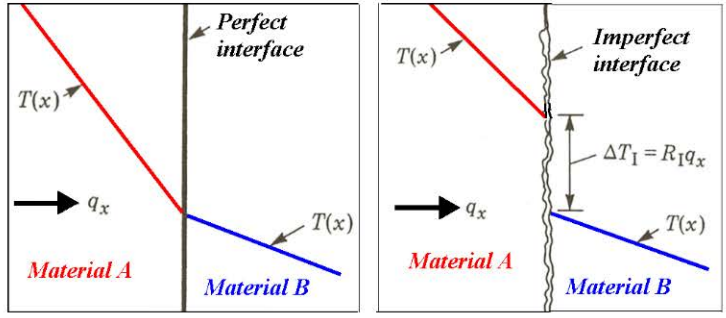


Fig. 129 Ideal and thermal contact resistance interfaces *[1]*

Contact Pressure	Moderate	100 kN/m^2	1e4 kN/m^2
Aluminum/aluminum/air	0.5	1.5-5.0	0.2-0.4
Copper/copper/air	0.1	1-10	0.1-0.5
Magnesium/magnesium/air	0.5	1.5-3.5	0.2-0.4
Stainless steel/stainless steel/ air	3	6-25	0.7-4.0

Fig. 130 Typical contact resistance values, R x e4, [m^2 K/W] *[1]*

Contacting Faces (pressure unknown)	Conductance
Aluminum / aluminum / air	2200 - 12000
Ceramic / ceramic / air	500 - 3000
Copper / copper / air	10,000 - 25,000
Iron / aluminum / air	45,000
Stainless steel / stainless steel / air	2000 - 3700
Stainless steel / stainless steel / vacuum	200 - 1100

Fig. 131 *Typical contact conductance values, C, [W/m^2 K] [1]*

The temperature often depends only on geometry. The heat flux, and the conthermal reaction, always depends on the material thermal conductivity. Therefore, it is always necessary to examine both the temperatures and heat flux to assure a correct solution. The heat flux is determined by the gradient (derivative) of the approximated temperatures. Therefore, it is less accurate than the temperatures. The user must make the mesh finer in regions where the heat flux vector is expected to rapidly change its value or direction. The heat flux should be plotted both as magnitude contours, and as vectors. The items available for output after a thermal analysis run are given in table from Fig. 132.

Symbol	Label	Item
T	TEMP	Temperature
$\partial T/\partial x$	GRADX	Temperature gradient in the selected reference X-direction
$\partial T/\partial y$	GRADY	Temperature gradient in the selected reference Y-direction
$\partial T/\partial z$	GRADZ	Temperature gradient in the selected reference Z-direction
$\lvert \nabla T \rvert$	GRADN	Resultant temperature gradient magnitude
q_x	HFLUXX	Heat flux in the X-direction of the selected reference geometry
q_y	HFLUXY	Heat flux in the X-direction of the selected reference geometry
q_z	HFLUXZ	Heat flux in the X-direction of the selected reference geometry
q	HFLUXN	Resultant heat flux magnitude

Fig. 132 Thermal analysis output options *[1]*

The temperatures should be plotted as discrete color bands or as contour lines. The temperature contours should be perpendicular to insulated boundaries. Near surfaces with specified temperatures, the contours should be nearly parallel to the surfaces. These "eyeball" checks are illustrated in Fig. 133.

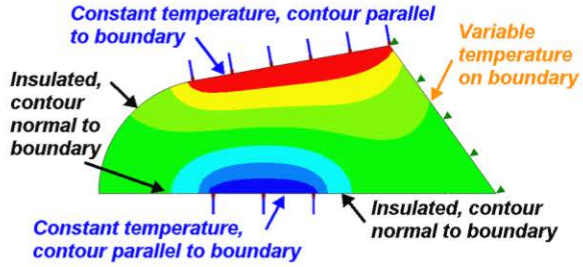


Fig. 133 *Guidelines for checking temperatures in isotropic materials [1]*

The heat flux vectors should be parallel to insulated surfaces. They should be nearly perpendicular to surfaces with a specified constant temperature. Those flux checks are illustrated in Fig. 134. These remarks on insulated boundaries do not apply if the material is anisotropic with the principal material directions inclined relative to the insulated surface (as will be seen later).

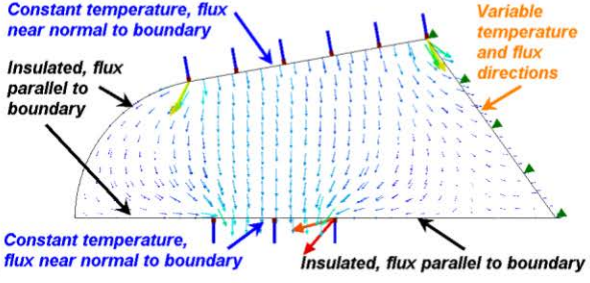


Fig. 134 Graphical checks for heat flux in isotropic materials *[1]*

The exact temperature gradient is discontinuous at an interface between different materials because their thermal conductivities will be different. Pretty continuous color contours (the default) tend to prevent these important engineering checks. The temperature and temperature

gradient vector can depend only on the geometry in some problems. Written results should not be given with more significant figures than the material input data. For heat transfer problems that is typically three or four significant figures.

In most analysis softwares it is possible to list, sum, average, and graph results along selected edges, lines, curves or surfaces. Thus, you should plan ahead and add "split lines" to the mesh where you expect to find such graphs informative. The thermal reaction heat flows is available in these application softwares while viewing the heat flux result plots.

4.5. Examples for solving commands

4.5.1. Solving the Model in CATIA

 (Compute) **Compute**: ↓ All, ↵ OK → **Computation Resources Estimation** , ↵ Yes → Computation Status.

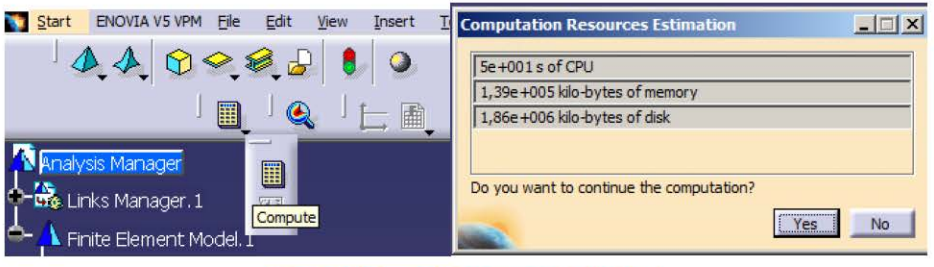


Fig. 135 Compute command in CATIA

4.5.2. Solving the Model in ANSYS

To solve the model click on the “Solve” button on the Standard Toolbar.

- Two processors used if present (default).
- To set the number of processors use, “Tools > Solve Process Settings”.

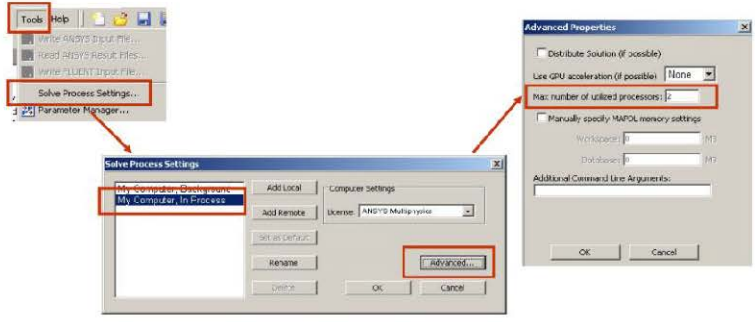


Fig. 136 Solving the model in ANSYS [6]