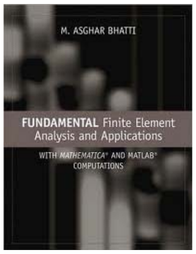
Finite Element Analysis of 3D Elastic Solids

Stresses and strains in elastic solids subjected to loading and temperature changes

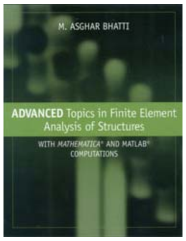
Displacements are small and the material behavior is linear elastic.

For further study

M.A. Bhatti
Fundamental Finite Element Concepts and Applications
with Computations using *Mathematica* and Matlab
John Wiley & Sons, New York, 2005



M.A. Bhatti
Advanced Topics in Finite Element Analysis of Structures
with Computations using *Mathematica* and Matlab
John Wiley & Sons, New York, 2006



John Wiley book web site

www.wiley.com/go/bhatti

Additional and more detailed examples

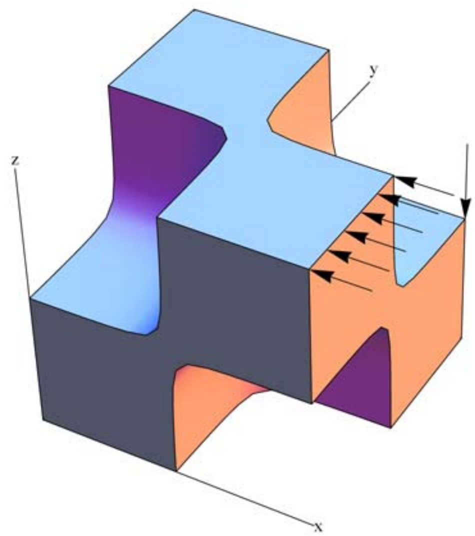
Mathematica, Matlab, and Ansys files for examples in the text

Partial answers to chapter end problems available from this web site

Three Dimensional Elasticity

Analysis assuming linear elastic material and small displacements

Consider an arbitrary solid supported in a stable manner under the influence of externally applied forces on its surface such as *pressure* and *concentrated loads*. In addition the solid may be subjected to applied forces distributed over the entire volume of the body. These forces are known as *body forces*. Typical examples of body forces are self weight of the object and centrifugal forces developed in rotating objects.



The governing differential equations can be developed easily by considering equilibrium of forces acting on an isolated part of the object. This approach gives three equilibrium equations in terms of stresses.

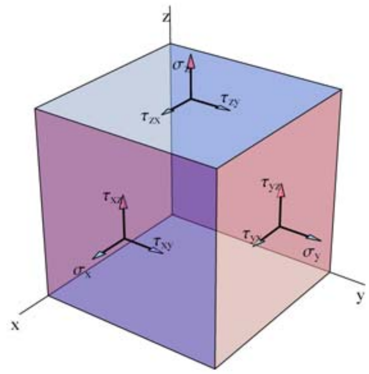
Considering equilibrium of forces acting on an isolated differential cube of the object the following differential equations can easily be derived.

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + b_x = 0$$

$$\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + b_y = 0$$

$$\frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + b_z = 0$$

where $(\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx})$ are the components of stresses on planes normal to the coordinate directions, $(\sigma_x, \sigma_y, \sigma_z)$ are the stresses normal to the planes and are called the normal stresses, $(\tau_{xy} = \tau_{yx}, \tau_{yz} = \tau_{zy}, \tau_{zx} = \tau_{xz})$ are the shear stresses on these planes, and (b_x, b_y, b_z) are components of body forces in the three coordinate directions. The body forces are part of applied forces that generate stresses in the body and are part of given data to start with a stress analysis.



The stresses are related to strains for different materials. For linear elastic isotropic materials the generalized Hook's law gives

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{pmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{pmatrix} \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix}$$

where E is the modulus of elasticity and ν is the Poisson's ratio.

The strains are related to displacements (u, v, w) along the three coordinate axes. Assuming small displacements it can be shown that

$$\begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} = \begin{pmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \end{pmatrix}$$

For an orthotropic material the constitutive matrix C has the following form

$$\boldsymbol{C} = \begin{pmatrix} \frac{E_x}{\gamma}\left(1 - \frac{E_z}{E_y}\nu_{yz}^2\right) & \frac{E_y \nu_{xy} + E_z \nu_{xz} \nu_{yz}}{\gamma} & \frac{E_z(\nu_{xz} + \nu_{xy}\nu_{yz})}{\gamma} & 0 & 0 & 0 \\ \frac{E_y \nu_{xy} + E_z \nu_{xz} \nu_{yz}}{\gamma} & \frac{E_y}{\gamma}\left(1 - \frac{E_z}{E_x}\nu_{xz}^2\right) & \frac{E_z}{\gamma}\left(\frac{E_y}{E_x}\nu_{xy}\nu_{xz} + \nu_{yz}\right) & 0 & 0 & 0 \\ \frac{E_z(\nu_{xz} + \nu_{xy}\nu_{yz})}{\gamma} & \frac{E_z}{\gamma}\left(\frac{E_y}{E_x}\nu_{xy}\nu_{xz} + \nu_{yz}\right) & \frac{E_z}{\gamma}\left(1 - \frac{E_y}{E_x}\nu_{xy}^2\right) & 0 & 0 & 0 \\ 0 & 0 & 0 & G_{xy} & 0 & 0 \\ 0 & 0 & 0 & 0 & G_{yz} & 0 \\ 0 & 0 & 0 & 0 & 0 & G_{xz} \end{pmatrix}$$

where E_x is Young's modulus in the x direction, ν_{xy} is Poisson's ratio relating ϵ_x to σ_y / E_y, etc and

$$\gamma = \frac{E_x\left(E_y - E_z\,\nu_{yz}^2\right) - E_y\left(E_y\,\nu_{xy}^2 + E_z\,\nu_{xz}\left(\nu_{xz} + 2\,\nu_{xy}\,\nu_{yz}\right)\right)}{E_x\,E_y}$$

$$G_{xy} = \frac{E_x}{2\left(1+\nu_{xy}\right)} \text{ etc.}$$

Presence of Initial Strains

Initial strain vector due to temperature change

$$\epsilon_0 = \begin{pmatrix} \alpha\,\Delta T \\ \alpha\,\Delta T \\ \alpha\,\Delta T \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

constitutive equations

$$\boldsymbol{\sigma} = \boldsymbol{C}\,(\boldsymbol{\epsilon} - \boldsymbol{\epsilon}_0)$$

Principal and Equivalent (von-Mises) Stresses

From a design point of view the six individual stress components usually are not useful. Material failure criteria is usually expressed in terms of principal stresses or some form of an equivalent stress.

Principal stresses

Writing the stress vectors as rows in a 3×3 matrix, the so-called *stress tensor* is defined as follows.

$$\boldsymbol{S} = \begin{pmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{pmatrix}$$

Principal directions are the unit normals for the planes over which there are no shear stresses and thus the normal stresses are maximum. Since the off-diagonal terms in the stress tensor ***S*** are the shear stresses, it follows that the principal planes are those that make the ***S*** matrix diagonal. Thus principal stresses can be determined by computing eigenvalues of matrix S.

The principal stresses are ordered according to their algebraic values:

$$\text{Maximum}: \sigma_1 \quad \text{Intermediate}: \sigma_2 \quad \text{Minimum}: \sigma_3$$

Class Activity

Stress components at a point in a solid are given as follows.

$$\sigma_x = 10; \quad \sigma_y = -7; \quad \sigma_z = 5; \quad \tau_{xy} = -3; \quad \tau_{xz} = 0; \quad \tau_{yz} = 2\,N/m^2$$

Compute principal stresses.

Solution

From the given stress components, the stress tensor is

```
S = {{10, -3, 0}, {-3, -7, 2}, {0, 2, 5}} // N
```

$$\begin{pmatrix} 10. & -3. & 0. \\ -3. & -7. & 2. \\ 0. & 2. & 5. \end{pmatrix}$$

The *Mathematica*'s Eigenvalues command is used to determine principal stresses.

```
ev = Eigenvalues[S]
```

{10.5353, −7.81721, 5.2819}

Ordering the eigenvalues from highest to lowest values, we have the following principal stresses and unit normals for the principal planes.

```
Sort[ev, Greater]
```

{10.5353, 5.2819, −7.81721}

Equivalent or von Mises stress

The *effective stress* or *von Mises stress* and is defined as follows

$$\sigma_e = \frac{1}{\sqrt{2}} \sqrt{\left((\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2\right)\right)}$$

For several engineering materials, particularly metals, this equivalent stress is compared to the uni-axial tensile strength of the material (σ_f) to determine safety factor.

Factor of safety = σ_f / σ_e

Class Activity

If a material has yield strength $\sigma_f = 50$ MPa, what is the factor of safety using the von Mises failure criterion if the state of stress at a point in a solid is given as follows.

$\sigma_x = 5$; $\sigma_y = -18$; $\sigma_z = 0$; $\tau_{xy} = 15$; $\tau_{xz} = 0$; $\tau_{yz} = 0$ MPa

Solution

```
sf = 50;
{sx, sy, sz, txy, tyz, tzx} = {5, -18, 0, 15, 0, 0};
```

```
se = Sqrt[(sx - sy)^2 + (sy - sz)^2 + (sz - sx)^2 + 6 (txy^2 + tyz^2 + tzx^2)] / Sqrt[2] // N
```

33.3766

```
FS = sf / se
```

1.49805

Potential energy

$$\Pi_p(u, v, w) = U - W$$

The strain energy is defined as follows.

$$U = \frac{1}{2} \iiint_V \boldsymbol{\epsilon}^T \boldsymbol{\sigma} \, \mathrm{dV}$$

where $\boldsymbol{\epsilon} = \left(\epsilon_x \quad \epsilon_y \quad \epsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right)^T$ is the strain vector, $\boldsymbol{\sigma} = \left(\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx} \right)^T$ is the stress vector

The work done term W incorporates work done by all applied forces, including body forces, distributed surface forces, and concentrated forces.

$$W_b = \iiint_V \left(b_x u + b_y v + b_z w \right) \mathrm{dV}$$

$$W_q = \iint_S \left(q_x u + q_y v + q_z w \right) \mathrm{dS}$$

$$W_f = \sum_i \left(F_{xi} u_i + F_{yi} v_i + F_{zi} w_i \right)$$

Minimum of potential energy corresponds to equilibrium equations. This is used in deriving finite element equations.

General Form of Finite Element Equations

Three unknowns: x, y, and z displacements

Each node in a finite element model has three degrees of freedom

Assumed solutions

$$u(x, y, z) = N_1 u_1 + N_2 u_2 + \ldots$$

$$v(x, y, z) = N_1 v_1 + N_2 v_2 + \ldots$$

$$w(x, y, z) = N_1 w_1 + N_2 w_2 + \ldots$$

$$\boldsymbol{u}(x, y, z) \equiv \begin{pmatrix} u \\ v \\ w \end{pmatrix} = \begin{pmatrix} N_1 & 0 & 0 & N_2 & 0 & \ldots \\ 0 & N_1 & 0 & 0 & N_2 & \ldots \\ 0 & 0 & N_1 & 0 & 0 & \ldots \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix} \equiv \boldsymbol{N}^T \boldsymbol{d}$$

From the assumed solution the element strain vector can be computed by appropriate differentiation as follows.

$$\boldsymbol{\epsilon} = \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} = \begin{pmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \end{pmatrix} = \begin{pmatrix} \frac{\partial N_1}{\partial x} & 0 & 0 & \frac{\partial N_2}{\partial x} & 0 & 0 & \ldots \\ 0 & \frac{\partial N_1}{\partial y} & 0 & 0 & \frac{\partial N_2}{\partial y} & 0 & \ldots \\ 0 & 0 & \frac{\partial N_1}{\partial z} & 0 & 0 & \frac{\partial N_2}{\partial z} & \ldots \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & 0 & \ldots \\ 0 & \frac{\partial N_1}{\partial z} & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial z} & \frac{\partial N_2}{\partial y} & \ldots \\ \frac{\partial N_1}{\partial z} & 0 & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial z} & 0 & \frac{\partial N_2}{\partial x} & \ldots \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix} \equiv \boldsymbol{B}^T \boldsymbol{d}$$

Using the constitutive matrix appropriate for the material the element stress vector can be written as follows.

$$\boldsymbol{\sigma} = \boldsymbol{C}\,\boldsymbol{\epsilon} = \boldsymbol{C}\,\boldsymbol{B}^T\,\boldsymbol{d}$$

Constitutive equations: For an isotropic material

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{pmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{pmatrix} \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix}$$

E = Modulus of elasticity and ν = Poisson's ratio

The strain energy over an element can now be written as follows.

$$U = \frac{1}{2} \iiint_V \boldsymbol{\epsilon}^T \boldsymbol{\sigma}\, \mathrm{dV} = \frac{1}{2} \iiint_V \left(\boldsymbol{B}^T \boldsymbol{d}\right)^T \boldsymbol{C}\,\boldsymbol{B}^T \boldsymbol{d}\, \mathrm{dV}$$

$$= \frac{1}{2} \boldsymbol{d}^T \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T\, \mathrm{dV}\, \boldsymbol{d} = \frac{1}{2} \boldsymbol{d}^T \boldsymbol{k}\; \boldsymbol{d}$$

where $\boldsymbol{k}$ is known as the element stiffness matrix

$$\boldsymbol{k} = \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T\, \mathrm{dV}$$

The work done by the body forces can be evaluated as follows.

$$W_b = \iiint_V \left(b_x\, u + b_y\, v + b_z\, w\right) \mathrm{dV} = \iiint_V \begin{pmatrix} b_x & b_y & b_z \end{pmatrix} \begin{pmatrix} u \\ v \\ w \end{pmatrix} \mathrm{dV}$$

Substituting the assumed solution we have

$$W_b = \iiint_V \begin{pmatrix} b_x & b_y & b_z \end{pmatrix} \boldsymbol{N}^T\, \mathrm{dV}\, \boldsymbol{d} \equiv \boldsymbol{r}_b^T\, \boldsymbol{d}$$

where $\boldsymbol{r}_b^T$ is the transpose of the equivalent nodal load vector due to body forces

$$\boldsymbol{r}_b = \iiint_V \boldsymbol{N} \begin{pmatrix} b_x \\ b_y \\ b_z \end{pmatrix} \mathrm{dV} = \begin{pmatrix} \iiint_V N_1\, b_x\, \mathrm{dV} \\ \iiint_V N_1\, b_y\, \mathrm{dV} \\ \iiint_V N_1\, b_z\, \mathrm{dV} \\ \iiint_V N_2\, b_x\, \mathrm{dV} \\ \vdots \end{pmatrix}$$

Assuming concentrated forces to be applied only at the nodes, the work done by the concentrated nodal forces can be evaluated as follows.

$$W_f = \sum_i \left(F_{\mathrm{xi}}\, u_i + F_{\mathrm{yi}}\, v_i + F_{\mathrm{zi}}\, w_i\right) = \begin{pmatrix} F_{\mathrm{x1}} & F_{\mathrm{y1}} & F_{\mathrm{z1}} & F_{\mathrm{x2}} & \ldots \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix} \equiv \boldsymbol{r}_f^T\, \boldsymbol{d}$$

where $\boldsymbol{r}_f^T$ is the transpose of the applied nodal load vector

$$\boldsymbol{r}_f = \begin{pmatrix} F_{x1} & F_{y1} & F_{z1} & F_{x2} & \ldots \end{pmatrix}^T$$

The work done by the applied surface forces:
Appropriate interpolation functions specific to the surface S are denoted by vector $\boldsymbol{N}_s$.

$$W_q = \iint_S (q_x\, u + q_y\, v + q_z\, w)\, \mathrm{dS} = \iint_S \begin{pmatrix} q_x & q_y & q_z \end{pmatrix} \begin{pmatrix} u \\ v \\ w \end{pmatrix} \mathrm{dS} = \iint_S \begin{pmatrix} q_x & q_y & q_z \end{pmatrix} \boldsymbol{N}_s^T\, \mathrm{dS}\, \boldsymbol{d} \equiv \boldsymbol{r}_q^T\, \boldsymbol{d}$$

$$\boldsymbol{r}_q = \iint_S \boldsymbol{N}_s \begin{pmatrix} q_x \\ q_y \\ q_z \end{pmatrix} \mathrm{dS} = \begin{pmatrix} \iint_S N_{s1}\, q_x\, \mathrm{dS} \\ \iint_S N_{s1}\, q_y\, \mathrm{dS} \\ \iint_S N_{s1}\, q_z\, \mathrm{dS} \\ \iint_S N_{s2}\, q_x\, \mathrm{dS} \\ \vdots \end{pmatrix}$$

$$\Pi_p(u, v, w) = U - W = \tfrac{1}{2}\, \boldsymbol{d}^T\, \boldsymbol{k}\, \boldsymbol{d} - \left(\boldsymbol{r}_q^T + \boldsymbol{r}_b^T + \boldsymbol{r}_f^T\right) \boldsymbol{d}$$

Using the necessary conditions for the minimum of potential energy are $\partial \Pi_p / \partial d_i = 0$

$$\boldsymbol{k}\; \boldsymbol{d} = \boldsymbol{r}_q + \boldsymbol{r}_b + \boldsymbol{r}_f$$

The concentrated forces applied at nodes can be assembled directly into the global load vector during assembly

$$\boldsymbol{k}\; \boldsymbol{d} = \boldsymbol{r}_q + \boldsymbol{r}_b$$

Finite Element Equations in the Presence of Initial Strains

Initial strain vector due to temperature change

$$\epsilon_0 = \begin{pmatrix} \alpha\, \Delta T \\ \alpha\, \Delta T \\ \alpha\, \Delta T \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Constitutive equations

$$\sigma = \boldsymbol{C}\, (\epsilon - \epsilon_0)$$

Strain energy

$$U = \tfrac{1}{2} \iiint_V (\epsilon - \epsilon_0)^T\, \boldsymbol{C}\, (\epsilon - \epsilon_0)\, \mathrm{dV}$$

Expanding the product

$$U = \frac{1}{2}\iiint_V \boldsymbol{\epsilon}^T \boldsymbol{C}\,\boldsymbol{\epsilon}\,\mathrm{dV} - \frac{1}{2}\iiint_V \boldsymbol{\epsilon}_0^T \boldsymbol{C}\,\boldsymbol{\epsilon}\,\mathrm{dV} - \frac{1}{2}\iiint_V \boldsymbol{\epsilon}^T \boldsymbol{C}\,\boldsymbol{\epsilon}_0\,\mathrm{dV} + \frac{1}{2}\iiint_V \boldsymbol{\epsilon}_0^T \boldsymbol{C}\,\boldsymbol{\epsilon}_0\,\mathrm{dV}$$

The last term will drop out when writing the necessary conditions for minimum.

$$U_e = \frac{1}{2}\iiint_V \boldsymbol{\epsilon}^T \boldsymbol{C}\,\boldsymbol{\epsilon}\,\mathrm{dV} - \iiint_V \boldsymbol{\epsilon}_0^T \boldsymbol{C}\,\boldsymbol{\epsilon}\,\mathrm{dV}$$

Substituting the strains in terms of the assumed solution $\left(\boldsymbol{\epsilon} = \boldsymbol{B}^T \boldsymbol{d}\right)$ we get

$$U_e = \frac{1}{2}\boldsymbol{d}^T \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T\,\mathrm{dV}\,\boldsymbol{d} - \iiint_V \boldsymbol{\epsilon}_0^T\,\boldsymbol{C}\,\boldsymbol{B}^T\,\mathrm{dV}\,\boldsymbol{d} = \frac{1}{2}\boldsymbol{d}^T\,\boldsymbol{k}\,\boldsymbol{d} - \boldsymbol{r}_\epsilon^T\,\boldsymbol{d}$$

where $\boldsymbol{k}$ is the usual element stiffness matrix

$$\boldsymbol{k} = \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T\,\mathrm{dV}$$

equivalent nodal load vector due to initial strains $\boldsymbol{\epsilon}_0$

$$\boldsymbol{r}_\epsilon = \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{\epsilon}_0\,\mathrm{dV}$$

General Form of Finite Element Equations with thermal strains

Assumed solutions

$$u(x, y, z) = N_1\,u_1 + N_2\,u_2 + \ldots$$

$$v(x, y, z) = N_1\,v_1 + N_2\,v_2 + \ldots$$

$$w(x, y, z) = N_1\,w_1 + N_2\,w_2 + \ldots$$

$$\boldsymbol{u}(x, y, z) \equiv \begin{pmatrix} u \\ v \\ w \end{pmatrix} = \begin{pmatrix} N_1 & 0 & 0 & N_2 & 0 & \ldots \\ 0 & N_1 & 0 & 0 & N_2 & \ldots \\ 0 & 0 & N_1 & 0 & 0 & \ldots \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix} \equiv \boldsymbol{N}^T\,\boldsymbol{d}$$

Strain displacement

$$\boldsymbol{\epsilon} = \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} = \begin{pmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \end{pmatrix} = \begin{pmatrix} \frac{\partial N_1}{\partial x} & 0 & 0 & \frac{\partial N_2}{\partial x} & 0 & 0 & \ldots \\ 0 & \frac{\partial N_1}{\partial y} & 0 & 0 & \frac{\partial N_2}{\partial y} & 0 & \ldots \\ 0 & 0 & \frac{\partial N_1}{\partial z} & 0 & 0 & \frac{\partial N_2}{\partial z} & \ldots \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & 0 & \ldots \\ 0 & \frac{\partial N_1}{\partial z} & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial z} & \frac{\partial N_2}{\partial y} & \ldots \\ \frac{\partial N_1}{\partial z} & 0 & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial z} & 0 & \frac{\partial N_2}{\partial x} & \ldots \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix} \equiv \boldsymbol{B}^T\,\boldsymbol{d}$$

Constitutive equations: For an isotropic material

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{pmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{pmatrix} \left(\begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} - \begin{pmatrix} \alpha\,\Delta T \\ \alpha\,\Delta T \\ \alpha\,\Delta T \\ 0 \\ 0 \\ 0 \end{pmatrix} \right)$$

$$\sigma = \boldsymbol{C}(\epsilon - \epsilon_0) = \boldsymbol{C}\,\boldsymbol{B}^T\,\boldsymbol{d} - \boldsymbol{C}\,\epsilon_0$$

Element equations

$$\boldsymbol{k}\,\boldsymbol{d} = \boldsymbol{r}_\epsilon + \boldsymbol{r}_q + \boldsymbol{r}_b$$

Element stiffness matrix: $\boldsymbol{k} = \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T\,dV$

Equivalent load vector due to temperature change: $\boldsymbol{r}_\epsilon = \iiint_V \boldsymbol{B}\,\boldsymbol{C}\,\epsilon_0\,dV$

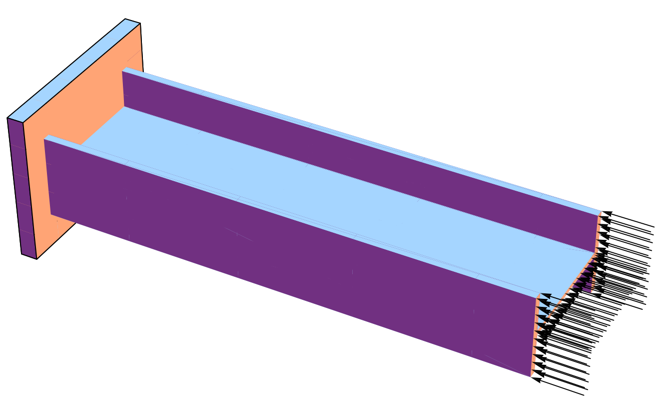
Equivalent load vector due to applied pressure: $\boldsymbol{r}_q = \iint_S \boldsymbol{N}\,\boldsymbol{q}\,dS$

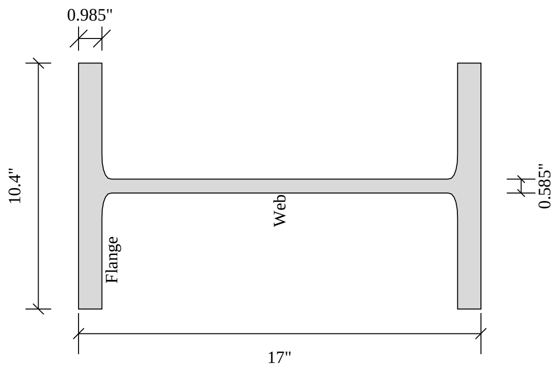
Equivalent load vector due to body forces: $\boldsymbol{r}_b = \iiint_V \boldsymbol{N}\,\boldsymbol{b}\,dV$

Finite Elements for 3D Analysis

Example

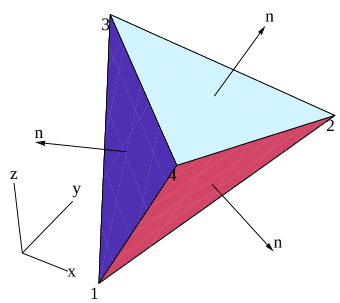
As a simple example to motivate discussion consider and I shaped solid fixed at one end and subjected to a uniform pressure of 10 ksi on the other as shown in the figure. The length of the member is 60 in and the cross section dimensions are as shown in the cross section. The fillets have a radius of 1.39 in. The material is steel with $E = 29\,000$ ksi and $\nu = 0.3$.





Tetrahedral Element

Four node tetrahedral element



Three degrees of freedom per node. The matrices for each element are 12×12.

Element stiffness matrix

$$\boldsymbol{k} = V\,\boldsymbol{B}\,\boldsymbol{C}\,\boldsymbol{B}^T$$

$$\boldsymbol{B}^T = \frac{1}{6V}\begin{pmatrix} b_1 & 0 & 0 & b_2 & 0 & 0 & \dots \\ 0 & c_1 & 0 & 0 & c_2 & 0 & \dots \\ 0 & 0 & d_1 & 0 & 0 & d_2 & \dots \\ c_1 & b_1 & 0 & c_2 & b_2 & 0 & \dots \\ 0 & d_1 & c_1 & 0 & d_2 & c_2 & \dots \\ d_1 & 0 & b_1 & d_2 & 0 & b_2 & \dots \end{pmatrix}\begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ \vdots \end{pmatrix}$$

V = volume of the element and $C = 6 \times 6$ material constitutive matrix representing relationship between stresses and strains. Constants $b_1, \dots, d_4$ are expressed in terms of nodal coordinates as follows.

$b_1 = y_4 (z_3 - z_2) + y_3 (z_2 - z_4) + y_2 (z_4 - z_3)$

$b_2 = y_4 (z_1 - z_3) + y_1 (z_3 - z_4) + y_3 (z_4 - z_1)$

$b_3 = y_4 (z_2 - z_1) + y_2 (z_1 - z_4) + y_1 (z_4 - z_2)$

$b_4 = y_3 (z_1 - z_2) + y_1 (z_2 - z_3) + y_2 (z_3 - z_1)$

$c_1 = x_4 (z_2 - z_3) + x_2 (z_3 - z_4) + x_3 (z_4 - z_2)$

$c_2 = x_4 (z_3 - z_1) + x_3 (z_1 - z_4) + x_1 (z_4 - z_3)$

$c_3 = x_4 (z_1 - z_2) + x_1 (z_2 - z_4) + x_2 (z_4 - z_1)$

$c_4 = x_3 (z_2 - z_1) + x_2 (z_1 - z_3) + x_1 (z_3 - z_2)$

$d_1 = x_4 (y_3 - y_2) + x_3 (y_2 - y_4) + x_2 (y_4 - y_3)$

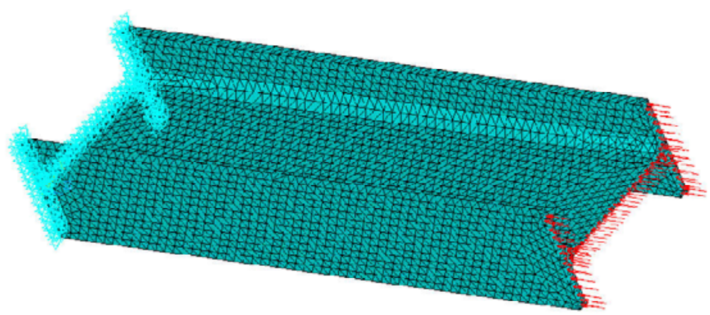
$d_2 = x_4 (y_1 - y_3) + x_1 (y_3 - y_4) + x_3 (y_4 - y_1)$

$d_3 = x_4 (y_2 - y_1) + x_2 (y_1 - y_4) + x_1 (y_4 - y_2)$

$d_4 = x_3 (y_1 - y_2) + x_1 (y_2 - y_3) + x_2 (y_3 - y_1)$

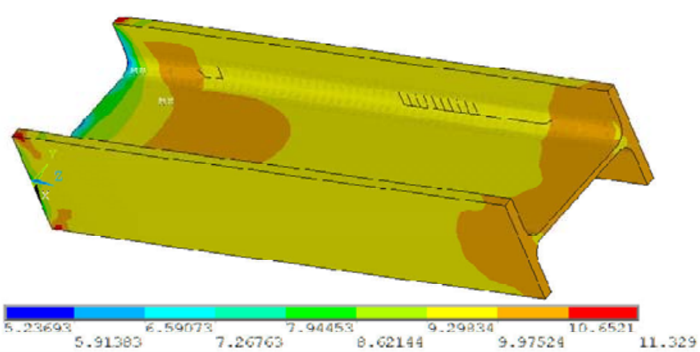
Ansys SOLID185

Model with 5187 nodes and 15358 tetrahedral elements



Tip displacement = 0.020466 in

von Mises stress (ksi) plot



Is a 3D solid model always necessary? Of course there are situations in which there is no other choice. However the answer in general is NO. We'll discuss several other models that are much simpler and may in fact give results that are easier to interpret.

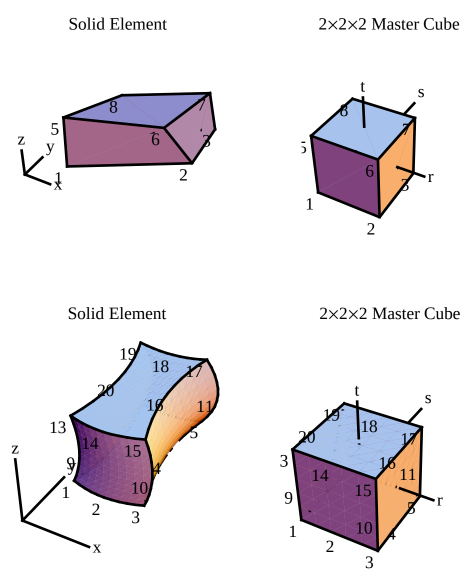
Solution

```
/PREP7
ET,1,SOLID185
MPTEMP,,,,,,,,
```

```
MPTEMP,1,0
MPDATA,EX,1,,29000
MPDATA,PRXY,1,,.3
*SET,d,17
*SET,tw,.585
*SET,bf,10.4
*SET,tf,0.985
*SET,kdes,1.39
K,1,-.5*bf,0,,
K,2,.5*bf,0,,
K,3,.5*bf,tf,,
K,4,.5*tw,tf,,
K,5,.5*tw,d-tf,,
K,6,.5*bf,d-tf,,
K,7,.5*bf,d,,
K,8,-.5*bf,d,,
K,9,-.5*bf,d-tf,,
K,10,-.5*tw,d-tf,,
K,11,-.5*tw,tf,,
K,12,-.5*bf,tf,,
LSTR,       1,       2
LSTR,       2,       3
LSTR,       3,       4
LSTR,       4,       5
LSTR,       5,       6
LSTR,       6,       7
LSTR,       7,       8
LSTR,       8,       9
LSTR,       9,      10
LSTR,      10,      11
LSTR,      11,      12
LSTR,      12,       1
LFILLT,4,3,kdes, ,
LFILLT,4,5,kdes, ,
LFILLT,9,10,kdes, ,
LFILLT,11,10,kdes, ,
FLST,2,16,4
FITEM,2,1
FITEM,2,2
FITEM,2,3
FITEM,2,13
FITEM,2,4
FITEM,2,14
FITEM,2,5
FITEM,2,6
FITEM,2,7
FITEM,2,8
FITEM,2,9
FITEM,2,15
FITEM,2,10
FITEM,2,16
FITEM,2,11
FITEM,2,12
AL,P51X
FLST,2,1,5,ORDE,1
FITEM,2,1
FLST,2,1,5,ORDE,1
FITEM,2,1
VEXT,P51X, , ,0,0,60,,,,
ESIZE,1,0,
MSHKEY,0
MSHAPE,1,3d
CM,_Y,VOLU
VSEL, , , ,       1
CM,_Y1,VOLU
```

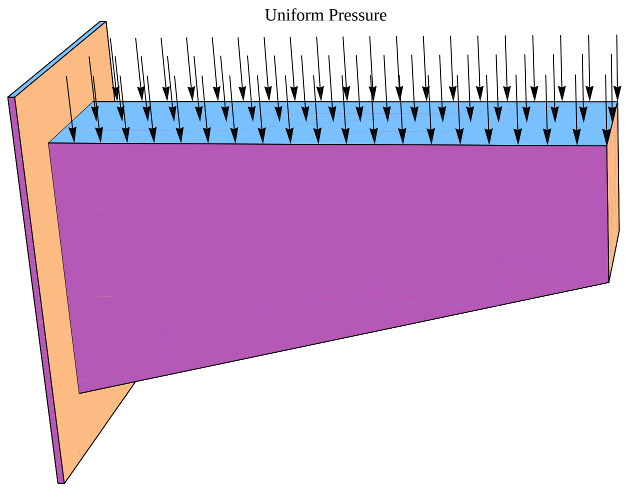
```
CHKMSH,'VOLU'
CMSEL,S,_Y
VCLEAR,_Y1
VMESH,_Y1
FLST,2,1,5,ORDE,1
FITEM,2,2
/GO
SFA,P51X,1,PRES,10
FLST,2,1,5,ORDE,1
FITEM,2,1
/GO
DA,P51X,ALL,
FINISH
/SOL
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLNSOL, S,EQV, 0,1.0
```

Mapped Solid Elements



Class Activity: Cantilever Bracket Using Tetrahedral Elements with Ansys

Use tetrahedral elements in Ansys (Solid185 with Tet meshing) to determine maximum von-Mises stress in the cantilever bracket shown in figure. The bracket is 400 mm long and 100 mm thick. At the base it is 200 mm deep and at the tip it is 100 mm. The top face is horizontal. The left end of the bracket is fixed. The top face is subjected to a pressure of 10MPa. The material properties are $E = 200$ GPa and $\nu = 0.3$.



Steps

- Preprocessor> Element Type> Add/Edit/Delete> Add> Structural Solid> Solid185
- Preprocessor> Material Props> Material Models> Material Model Number 1> Structural> Linear> Elastic> Isotropic> EX [200000 N/mm^2] and PRXY [0.3]
- Material> Exit

The geometry can easily be created by first defining the front elevation face and then extruding it through the thickness. To define the front face create 4 key points. With the origin at the left end bottom the key points can be created by using the following menu path.

You can define variables and use them in creating finite element model. It is not necessary to do it for a standard finite element analysis. However for use later in optimization you must use variables for items that you want to treat as design variables. In preparation for that define the following two parameters for y coordinates of the bottom face of the bracket.

Parameters> Scalar parameters. In the resulting box in the line below Selection enter a = 0 and click Accept. It should show a parameter A = 0 in the larger area. Similarly define b = 100.

- Preprocessor> Modeling> Create> Keypoints> In the active CS (Enter keypoint # and x, y, z coordinates)

Parameters a and b were defined for y coordinates of key points 1 and 2 respectively. Use the parameters instead of numbers when defining the coordinates of these key points.

Key point 1: 0, a, 0

Key point 2: 400, b, 0

Key point 3: 400, 200, 0

Key point 4: 0, 200, 0

After defining all key points the area can be created by using the following menu path and picking the keypoints.

- Preprocessor> Modeling> Create> Areas> Arbitrary> Through KPs [Pick keypoints while moving counterclockwise to define the front face]
- Preprocessor> Modeling> Operate> >Extrude> Areas> By XYZ Offset. Pick area and enter [dx, dy, dz] = [0, 0, 100] to create the solid.

    PlotCtrls> Pan, Zoom, Rotate> iso to see 3D view.

The next step is to actually create a finite element mesh. We must decide on an approximate size of each element. Looking at the physical dimensions of the model, we choose a global element size of 50 mm for a coarse mesh.

- Preprocessor> Meshing> Size Cntrls> ManualSize> Global> Size [50]
- Preprocessor> Meshing> Mesh> Areas> Free [Pick area]

The final task before a solution can be initiated is to specify boundary conditions. The fixed end condition on right end is specified by constraining all nodes on line L4 as follows.

- Solution> Loads> Define Loads> Apply> Structural> Displacement> On areas. Select the supported face of the bracket and set all degrees of freedom to 0. You may hae to rotate the solid to bring the appropriate face in view to select it.

    PlotCtrls> Pan, Zoom, Rotate

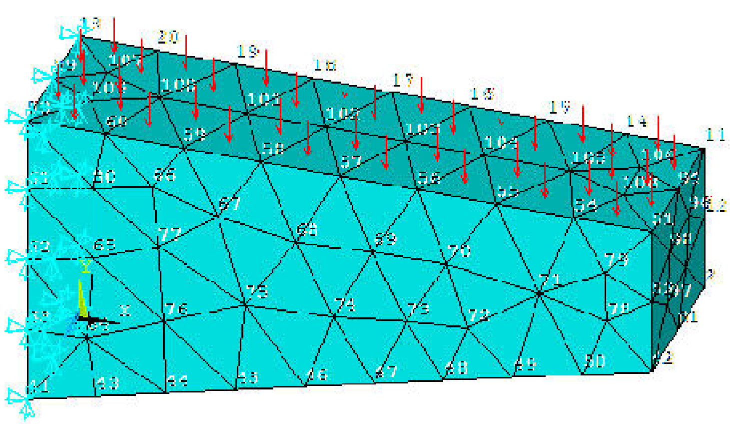
- Solution> Loads> Define Loads> Apply> Structural> Pressure> On areas. Pick the top face and enter pressure 10 N/mm^2.

PlotCtrls> Numbering. Select appropriate options to show node number, key point numbers, etc on the plots

PlotCtrls> Symbols. Select options to show load and boundary conditions symbols on plts. To see pressure as arrows in the/PSF section set ‘Show pressure and convect’ as ‘Arrows’.

PlotCtrls> Hard copy> To File. Select desired file type and filename to save graphics window contents to a file.

Model showing loads, boundary conditions, and node numbers



We are now ready to actually perform the finite element analysis that is done by using the following menus.

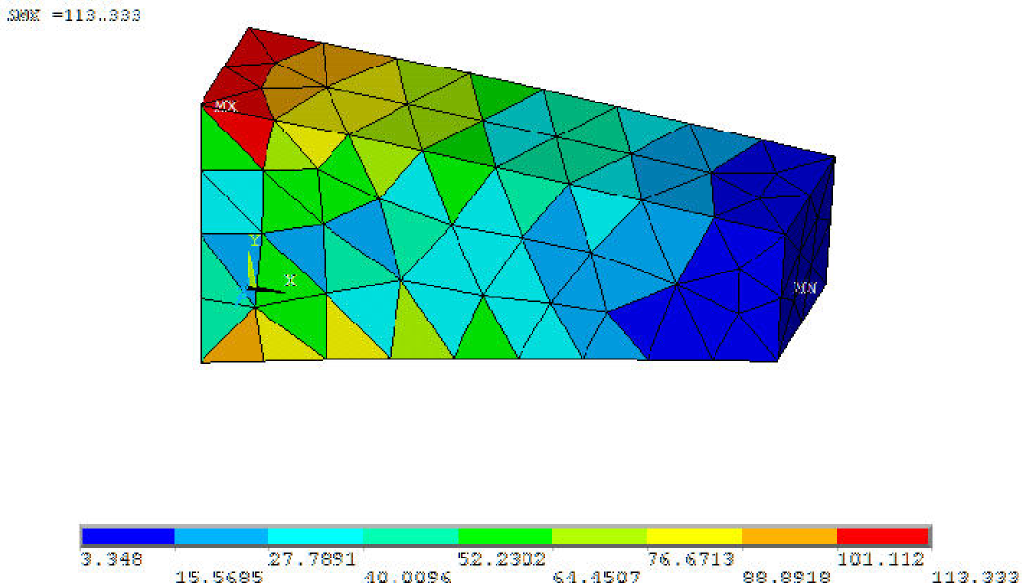
- Solution> Analysis Type> New Analysis> [Pick Static analysis (default)]
- Solution> Solve> Current LS

After the solution is done the results are viewed using the general postprocessor. First the results are read from the database.

- General postproc> Postprocessor> Read results> First set

The results can now be viewed as numerical lists or plotted in various forms. For example Figure A.8 shows a contour plot of equivalent von Mises stresses obtained using the following menu path.

- General postproc> Plot Results> Contour Plot> Element Soln> Stress> von Mises SEQV



Solution

The stress over the element is constant. Note that the stresses at the same node from two different elements are very different. This indicates that the mesh is very coarse and must be refined for a reasonable solution.

The complete set of equivalent text commands to solve this example is as follows.

Definition of parameters needed for optimization

General Postproc> Element Table> Define Table. In the resulting dialog box click Add. From the resulting dialog box pick Geometry and then Elem volume VOLU. Enter a desired label such as evol. This defines a table of element volumes. It is used in the next step to define a variable for the total volume of the model.

Parameters> Get Scalar Data> Results data> Element table sums> OK. In the resulting dialog box enter a label of your choice (say volume) and pick the element table item (evol) defined in the previous step. This parameter will be used later to define total volume of the model as an objective function.

Parameters> Get Scalar Data> Results data> Nodal results> OK. In the resulting dialog box enter a label of your choice, a node number, pick DOF solution, and component. For the example we'll enter label disp, node number 94, and UZ component. This will be used later to define a tip displacement constraint.

Parameters> Get Scalar Data> Results data> Nodal results> OK. In the resulting dialog box enter a label of your choice, a node number, pick Stress, and component. For the example we'll enter label strs, node number 99, and von Mises stress. This will be used later to define a stress constraint at the base of the cantilever.

Parameters> Scalar parameters. Make sure that you see all parameters defined in the box with values that make sense.

Click on the Session Editor (Second to last command in the Main Menu). Highlight and copy everything including and below /PREP7 line. Use a text editor and paste these commands in a new file. Save this file (with an extension .txt).

Optimization problem setup

Design Opt> Analysis file> Assign. Browse to locate the file created above and make sure its full path-name is entered.

Design Opt> Design variables. Click Add from the resulting dialog box. Pick parameter A as your first design variable. Enter a minimum and maximum value for it. You can leave the tolerance blank. For this example we'll set the range of this variable to (0, 100).

Similarly set B as second variable with range (100, 150).

Design Opt> State variables. Click Add from the resulting dialog box. Pick parameter 'disp' as your first constraint. Enter a minimum and maximum value for it. You can leave the tolerance blank. For this example since the tip displacements are negative, we'll enter a lower limit of -0.002 mm as our displacement constraint.

Similarly set strs as second constraint with maximum value as 150 N/mm^2.

Design Opt> Objective. Pick parameter 'volume' as your objective function.

Design Opt> Method/Tool> Random designs. Set iterations to 5.

Design Opt> Run.

Design Opt> Design sets> List> All sets.

```
LIST OPTIMIZATION SETS FROM SET   1 TO SET  13 AND SHOW
ONLY OPTIMIZATION PARAMETERS. (A "*" SYMBOL IS USED TO
INDICATE THE BEST LISTED SET)

                 SET   1        SET   2        SET   3        SET   4
               (INFEASIBLE)   (FEASIBLE)     (FEASIBLE)     (INFEASIBLE)
DISP    (SV)   -0.15709E-02    0.0000         0.27614E-03  >-0.36551E-02
STRS    (SV)     107.77        139.86         113.89          93.435
A       (DV)  >  0.0000        84.733         38.861          53.802
B       (DV)     100.00        122.86         134.62          130.90
VOLUME  (OBJ)   0.60000E+07   0.38482E+07    0.45304E+07    0.43060E+07

                 SET   5        SET   6        SET   7        SET   8
               (FEASIBLE)     (FEASIBLE)     (FEASIBLE)     (FEASIBLE)
DISP    (SV)   -0.17386E-02    0.0000         0.0000         0.60102E-02
STRS    (SV)     126.32        34.467         66.817          10.476
A       (DV)     84.800        46.963         79.294          69.545
B       (DV)     100.17        116.01         129.11          103.50
VOLUME  (OBJ)   0.43007E+07   0.47405E+07    0.38320E+07    0.45391E+07
```

Try other optimization methods

Design Opt> Method/Tool> Gradient

```
LIST OPTIMIZATION SETS FROM SET   1 TO SET  13 AND SHOW
ONLY OPTIMIZATION PARAMETERS. (A "*" SYMBOL IS USED TO
INDICATE THE BEST LISTED SET)

                 SET   9        SET  10        SET  11       *SET 12*
```

		(INFEASIBLE)	(FEASIBLE)	(FEASIBLE)	(FEASIBLE)
DISP	(SV)	>-0.84055E-02	0.0000	-0.34106E-04	0.0000
STRS	(SV)	104.59	93.856	60.532	150.53
A	(DV)	34.793	53.021	63.204	79.793
B	(DV)	105.62	149.47	107.54	129.11
VOLUME	(OBJ)	0.51917E+07	0.39501E+07	0.45850E+07	0.38220E+07

		SET 13
		(FEASIBLE)
DISP	(SV)	0.0000
STRS	(SV)	149.68
A	(DV)	79.294
B	(DV)	129.36
VOLUME	(OBJ)	0.38270E+07