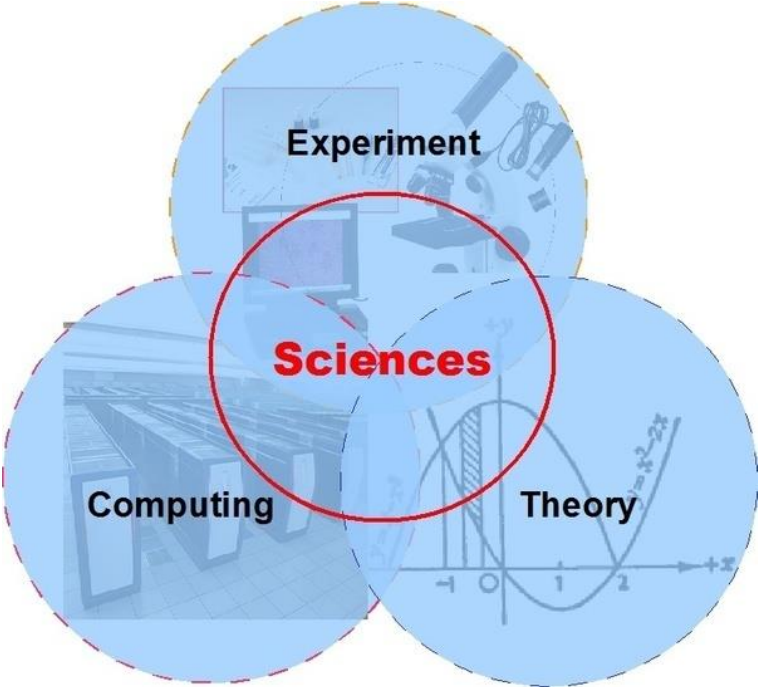
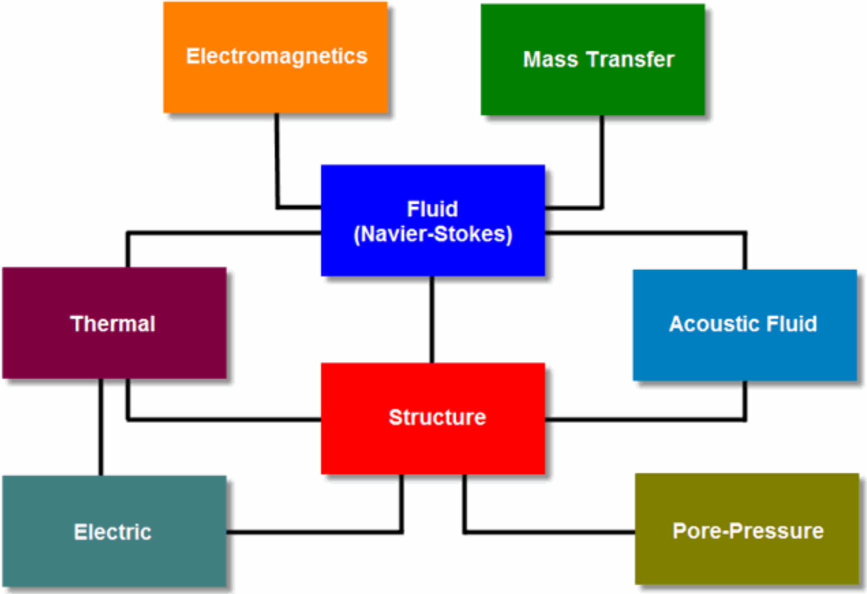
MODELOWANIE I OBLICZENIA W BADANIACH NAUKOWYCH

Tomasz Strek
Institute of Applied Mechanics,
Poznan University of Technology
ul. Jana Pawla II 24, 60-965 Poznan, Poland
Room 438
www.strek.h2g.pl

DATE: 2020.03.02


Experiment
Sciences
Computing
Theory


Electromagnetics
Mass Transfer
Fluid
(Navier-Stokes)
Thermal
Acoustic Fluid
Structure
Electric
Pore-Pressure

Solving Time-Dependent Problems

To get the equation for a time-dependent PDE in COMSOL Multiphysics, add terms containing time derivatives to the left-hand side of the stationary equation. The time derivatives must appear linearly, and the Dirichlet conditions must be linear. A time-dependent problem in the coefficient form reads

$$\begin{cases} e_a \dfrac{\partial^2 u}{\partial t^2} + d_a \dfrac{\partial u}{\partial t} + \nabla \cdot (-c\nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f & \text{in } \Omega \\ \mathbf{n} \cdot (c\nabla u + \alpha u - \gamma) + qu = g - h^T \mu & \text{on } \partial\Omega \\ hu = r & \text{on } \partial\Omega \end{cases}$$

THE SCALAR COEFFICIENT FORM EQUATION

A single dependent variable u is an unknown function on the computational domain. COMSOL Multiphysics determines it by solving the PDE problem that you specify. In coefficient form, the PDE problem reads

$$\begin{cases} e_a \dfrac{\partial^2 u}{\partial t^2} + d_a \dfrac{\partial u}{\partial t} + \nabla \cdot (-c\nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f & \text{in } \Omega \\ \mathbf{n} \cdot (c\nabla u + \alpha u - \gamma) + qu = g - h^T \mu & \text{on } \partial\Omega \\ hu = r & \text{on } \partial\Omega \end{cases} \tag{9-1}$$

where

- Ω is the computational domain—the union of all subdomains
- $\partial\Omega$ is the domain boundary
- $\mathbf{n}$ is the outward unit normal vector on $\partial\Omega$

The first equation in the list above is the PDE, which must be satisfied in Ω. The second and third equations are the boundary conditions, which must hold on $\partial\Omega$. The second equation is a *generalized Neumann* boundary condition, whereas the third equation is a *Dirichlet* boundary condition. This nomenclature and the second equation above deviate slightly from traditional usage in potential theory where a Neumann condition usually refers to the case $q = 0$. The generalized Neumann condition is also called a *mixed boundary condition* or a *Robin boundary condition*. In finite element terminology, Neumann boundary conditions are called *natural boundary conditions* because they do not occur explicitly in the weak form of the PDE problem. Dirichlet conditions are called *essential boundary conditions* because they restrict the trial space. Dirichlet boundary conditions often represent *constraints*.

- The symbol ∇ is the vector differential operator (gradient), defined as

$$\nabla = \left(\frac{\partial}{\partial x_1}, \dots, \frac{\partial}{\partial x_n}\right)$$

  The space coordinates are denoted $x_1, \dots, x_n$, where n represents the number of space dimensions.

- The symbol Δ stands for the Laplace operator

- The symbol Δ stands for the Laplace operator

$$\frac{\partial^2}{\partial x_1^2} + \ldots + \frac{\partial^2}{\partial x_n^2}$$

- $\nabla \cdot (c \nabla u)$ means

$$\frac{\partial}{\partial x_1}\left(c \frac{\partial u}{\partial x_1}\right) + \ldots + \frac{\partial}{\partial x_n}\left(c \frac{\partial u}{\partial x_n}\right)$$

- $\beta \cdot \nabla u$ means

$$\beta_1 \frac{\partial u}{\partial x_1} + \ldots + \beta_n \frac{\partial u}{\partial x_n}$$

where $\beta_1, \ldots, \beta_n$ are the components of the vector β.

THE COEFFICIENT FORM EQUATION SYSTEM

With two independent variables u_1 and u_2, the stationary PDE problem in coefficient form results in the following equation system:

$$e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} - \nabla \cdot (c\nabla u + \alpha u - \gamma) + \beta \cdot \nabla u + au = f$$

where $u = (u_1, u_2)$. The mass term is defined as

$$e_a \frac{\partial^2 u}{\partial t^2} = \begin{bmatrix} e_{a11} & e_{a12} \\ e_{a21} & e_{a22} \end{bmatrix} \begin{bmatrix} \frac{\partial^2 u_1}{\partial t^2} \\ \frac{\partial^2 u_2}{\partial t^2} \end{bmatrix} = \begin{bmatrix} e_{a11} \frac{\partial^2 u_1}{\partial t^2} + e_{a12} \frac{\partial^2 u_2}{\partial t^2} \\ e_{a21} \frac{\partial^2 u_1}{\partial t^2} + e_{a22} \frac{\partial^2 u_2}{\partial t^2} \end{bmatrix}$$

Similarly, the damping term is

$$d_a \frac{\partial u}{\partial t} = \begin{bmatrix} d_{a11} & d_{a12} \\ d_{a21} & d_{a22} \end{bmatrix} \begin{bmatrix} \frac{\partial u_1}{\partial t} \\ \frac{\partial u_2}{\partial t} \end{bmatrix} = \begin{bmatrix} d_{a11} \frac{\partial u_1}{\partial t} + d_{a12} \frac{\partial u_2}{\partial t} \\ d_{a21} \frac{\partial u_1}{\partial t} + d_{a22} \frac{\partial u_2}{\partial t} \end{bmatrix}$$

However, if $e_a = 0$, then d_a is often called the mass coefficient.

Solving Time-Dependent Problems

To get the equation for a time-dependent PDE in COMSOL Multiphysics, add terms containing time derivatives to the left-hand side of the stationary equation. The time derivatives must appear linearly, and the Dirichlet conditions must be linear. A time-dependent problem in the coefficient form reads

$$\begin{cases} e_a \dfrac{\partial^2 u}{\partial t^2} + d_a \dfrac{\partial u}{\partial t} + \nabla \cdot (-c\nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f & \text{in } \Omega \\ \mathbf{n} \cdot (c\nabla u + \alpha u - \gamma) + qu = g - h^T \mu & \text{on } \partial\Omega \\ hu = r & \text{on } \partial\Omega \end{cases}$$

If u is a vector of dependent variables then the *mass coefficient* e_a is a matrix. All coefficients can depend on time. The name *mass matrix* or mass coefficient stems from the fact that in many physics applications e_a contains the mass density. The d_a coefficient represents damping for wave-type equations. However, if $e_a = 0$, then d_a is often called the mass coefficient. The default settings are $e_a = 0$ and $d_a = 1$, representing a time-dependent PDE such as the heat equation. Using $e_a = 1$ and $d_a = 0$ represents an undamped wave equation.

The time-dependent problem in the general form is

$$\begin{cases} e_a \dfrac{\partial^2 u}{\partial t^2} + d_a \dfrac{\partial u}{\partial t} + \nabla \cdot \Gamma = F & \text{in } \Omega \\ -\mathbf{n} \cdot \Gamma = G + \left(\dfrac{\partial R}{\partial u}\right)^T \mu & \text{on } \partial\Omega \\ 0 = R & \text{on } \partial\Omega \end{cases}$$

The flux vector Γ and the scalar coefficients F, G, and R can be functions of the spatial coordinates, the solution u, and the space and time derivatives of u. The superscript "T" in the Neumann boundary condition denotes the transpose. The variable μ is the Lagrange multiplier.

Time-Dependent Systems

For time-dependent systems of PDEs the d_a and e_a coefficients are matrices. For example, for the system

$$e_{11}\frac{\partial^2 u_1}{\partial t^2}+e_{12}\frac{\partial^2 u_2}{\partial t^2}+d_{11}\frac{\partial u_1}{\partial t}+d_{12}\frac{\partial u_2}{\partial t}+\nabla\cdot(-c_{11}\nabla u_1-c_{12}\nabla u_2)+a_{11}u_1+a_{12}u_2 = f_1$$

$$e_{21}\frac{\partial^2 u_1}{\partial t^2}+e_{22}\frac{\partial^2 u_2}{\partial t^2}+d_{21}\frac{\partial u_1}{\partial t}+d_{22}\frac{\partial u_2}{\partial t}+\nabla\cdot(-c_{21}\nabla u_1-c_{22}\nabla u_2)+a_{21}u_1+a_{22}u_2 = f_2$$

the coefficient matrices are

$$e_a = \begin{bmatrix} e_{11} & e_{12} \\ e_{21} & e_{22} \end{bmatrix} \qquad d_a = \begin{bmatrix} d_{11} & d_{12} \\ d_{21} & d_{22} \end{bmatrix}$$

$$c = \begin{bmatrix} c_{11} & c_{12} \\ c_{21} & c_{22} \end{bmatrix} \qquad a = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}$$

(where each element c_{ij} can be an n-by-n matrix). Many interesting problems have a singular d_a matrix (with $e_a = 0$), or a singular nonzero e_a matrix. Such problems are

Solving Eigenvalue Problems

COMSOL Multiphysics handles scalar eigenvalue problems for all PDE forms. These eigenvalue problems are related to time-dependent problems via the correspondence $\partial/\partial t \leftrightarrow -\lambda$, linking the time derivative to the eigenvalue λ (with the default eigenvalue name `lambda`). An eigenvalue problem in the coefficient form reads

$$\begin{cases} (\lambda-\lambda_0)^2 e_a u - (\lambda-\lambda_0) d_a u + \nabla\cdot(-c\nabla u - \alpha u) + \beta\cdot\nabla u + au = f & \text{in } \Omega \\ \mathbf{n}\cdot(c\nabla u + \alpha u) + qu = -h^T\mu & \text{on } \partial\Omega \\ hu = r & \text{on } \partial\Omega \end{cases}$$

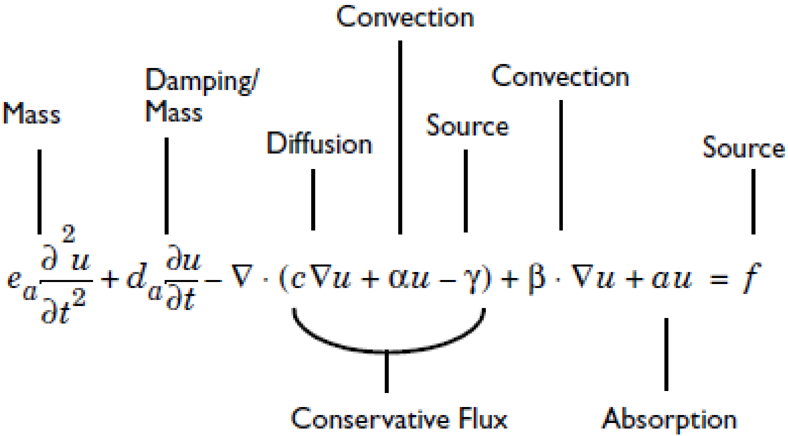
where λ_0 is the linearization point for the eigenvalue. Note that the source terms are ignored if the solution form is coefficient form. If the general or weak solution forms are used, the source terms are not ignored if they depend on the solution components. If the coefficients depend on u or the eigenvalue λ, COMSOL Multiphysics performs a linearization of the problem about the linearization point $u = u_0$, $\lambda = \lambda_0$. The software also performs this linearization for eigenvalue problems in the general and weak forms, though in a slightly different way. See “The Linear or Linearized Model” on page 386 in the *COMSOL Multiphysics User’s Guide* for information about linearization.

There are many interesting PDE problems to which this interpretation does not apply. For instance, a time-harmonic PDE such as the Helmholtz equation represents a time-dependent phenomenon transformed into the frequency domain.

For the Neumann boundary condition of the coefficient form

$$\mathbf{n} \cdot (c\nabla u + \alpha u - \gamma) + qu = g - h^T \mu$$

- q is the *boundary absorption coefficient.*
- g is the *boundary source term.*


Convection
Damping/ Mass
Mass
Diffusion
Source
Convection
Source
$e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} - \nabla \cdot (c\nabla u + \alpha u - \gamma) + \beta \cdot \nabla u + a u = f$
Conservative Flux
Absorption

Interpreting PDE Coefficients

The COMSOL PDE formulations can model a variety of problems, but note that this documentation uses coefficient names that fall within the realm of continuum mechanics and mass transfer. For the coefficient form:

- e_a is the *mass coefficient.*
- d_a is a *damping coefficient* or a *mass coefficient.*
- c is the *diffusion coefficient.*
- α is the *conservative flux convection coefficient.*
- β is the *convection coefficient.*
- a is the *absorption coefficient.*
- γ is the *conservative flux source term.*
- f is the *source term.*

TABLE 9-2: CLASSICAL PDES IN COMPACT AND STANDARD NOTATION

EQUATION	COMPACT NOTATION	STANDARD NOTATION (2D)
Laplace's equation	$-\nabla\cdot(\nabla u) = 0$	$-\frac{\partial}{\partial x}\frac{\partial u}{\partial x}-\frac{\partial}{\partial y}\frac{\partial u}{\partial y} = 0$
Poisson's equation	$-\nabla\cdot(c\nabla u) = f$	$-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right) = f$
Helmholtz equation	$-\nabla\cdot(c\nabla u)+au = f$	$-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right)+au = f$
Heat equation	$d_a\frac{\partial u}{\partial t}-\nabla\cdot(c\nabla u) = f$	$d_a\frac{\partial u}{\partial t}-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right) = f$
Wave equation	$e_a\frac{\partial^2 u}{\partial t^2}-\nabla\cdot(c\nabla u) = f$	$e_a\frac{\partial^2 u}{\partial t^2}-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right) = f$
Schrödinger equation	$-\nabla\cdot(c\nabla u)+au = \lambda u$	$-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right)+au = \lambda u$
Convection-diffusion equation	$d_a\frac{\partial u}{\partial t}-\nabla\cdot(c\nabla u)+\beta\cdot\nabla u = f$	$d_a\frac{\partial u}{\partial t}-\frac{\partial}{\partial x}\left(c\frac{\partial u}{\partial x}\right)-\frac{\partial}{\partial y}\left(c\frac{\partial u}{\partial y}\right)$ $+\beta_x\frac{\partial u}{\partial x}+\beta_y\frac{\partial u}{\partial y} = f$

Recall the coefficient form system of equations

$$\begin{cases} e_{a\,lk}\dfrac{\partial^2 u_k}{\partial t^2} + d_{a\,lk}\dfrac{\partial u_k}{\partial t} + \nabla\cdot(-c_{lk}\nabla u_k - \alpha_{lk}u_k + \gamma_l) + \beta_{lk}\cdot\nabla u_k + a_{lk}u_k = f_l & \text{in } \Omega \\ \mathbf{n}\cdot(c_{lk}\nabla u_k + \alpha_{lk}u_k - \gamma_l) + q_{lk}u_k = g_l - h_{ml}\mu_m & \text{on } \partial\Omega \\ h_{mk}u_k = r_m & \text{on } \partial\Omega \end{cases}$$

where k and l range from 1 to N, and m ranges from 1 to M, where $M \le N$. Let y be the name of space coordinate number j. The notation in the following table uses the summation convention; that is, there is an implicit sum over all pairs of equal indices.

Poisson's Equation on the Unit Disk

A classic PDE with a well-known behavior is Poisson's equation

$$\begin{cases} -\nabla \cdot (\nabla u) = f & \Omega \\ u = 0 & \partial\Omega \end{cases}$$

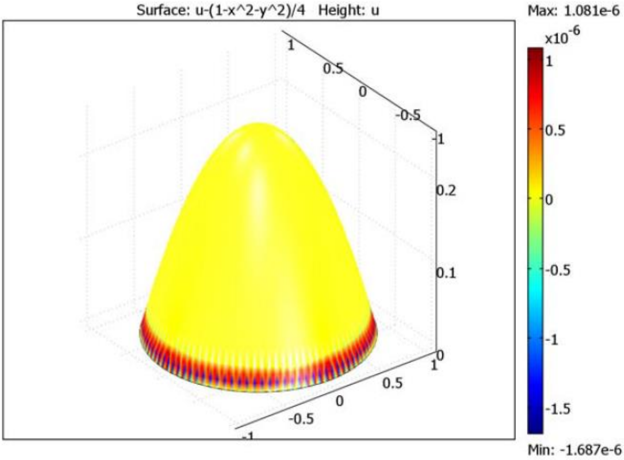
on the unit disk Ω with $f = 1$. The exact solution is

$$u(x, y) = \frac{1 - x^2 - y^2}{4},$$

which makes it possible to compare the COMSOL Multiphysics solution with the values of the exact solution at the node points on the mesh.

Results

The plot below shows the error (the difference between the numeric solution in COMSOL Multiphysics and the exact, analytic solution):



Fundamentals of Acoustics

What is Acoustics?

Acoustics is the physics of *sound*. Sound is the sensation, as detected by the ear, of very small rapid changes in the air pressure above and below a static value. This static value is atmospheric pressure (about 100,000 pascals), which varies slowly. Associated with a sound pressure wave is a flow of energy. Physically, sound in air is a longitudinal wave where the wave motion is in the direction of the energy flow. The wave crests are the pressure maxima, while the troughs represent the pressure minima.

Sound results when the air is disturbed by some source. An example is a vibrating object, such as a speaker cone in a hi-fi system. It is possible to see the movement of a bass speaker cone when it generates sound at a very low frequency. As the cone moves forward, it compresses the air in front of it, causing an increase in air pressure. Then it moves back past its resting position and causes a reduction in air pressure. This process continues, radiating a wave of alternating high and low pressure at the speed of sound.

Mathematical Models for Acoustic Analysis

Sound waves in a lossless medium are governed by the following equation for the (differential) pressure p (with SI unit N/m^2):

$$\frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = Q$$

Here ρ_0 (kg/m^3) refers to the density and c_s (m/s) denotes the speed of sound. The *dipole source* $\mathbf{q}$ (N/m^3) and the *monopole source* Q ($1/s^2$) are both optional. The combination $\rho_0 c_s^2$ is called the *bulk modulus*, commonly denoted β (N/m^2).

A special case is a time-harmonic wave, for which the pressure varies with time as

$$p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$$

where $\omega = 2\pi f$ (rad/s) is the angular frequency, f (Hz) as usual denoting the frequency. Assuming the same harmonic time-dependence for the source terms, the wave equation for acoustic waves reduces to an inhomogeneous Helmholtz equation:

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = Q \qquad (3\text{-}1)$$

With the source terms removed, you can also treat this equation as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies.

Typical boundary conditions are:

- Sound-hard boundaries (walls)
- Sound-soft boundaries
- Impedance boundary conditions
- Radiation boundary conditions

In lossy media, an additional term of first order in the time derivative needs to be introduced to model attenuation of the sound waves:

$$\frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} - d_a \frac{\partial p}{\partial t} + \nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = Q$$

TIME-HARMONIC ANALYSIS

The time-harmonic—or frequency-domain—formulation is based on the inhomogeneous Helmholtz equation given in Equation 3-1 on page 21 and repeated here for convenience:

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = Q$$

With this formulation you can compute the frequency response using the parametric solver to sweep over a frequency range using a harmonic load.

EIGENFREQUENCY ANALYSIS

In the eigenfrequency formulation the source terms are absent and you solve for the eigenmodes and the eigenvalues or eigenfrequencies:

$$\nabla \cdot \left(-\frac{1}{\rho_0}\nabla p\right) + \frac{\lambda^2 p}{\rho_0 c_s^2} = 0 \qquad (3\text{-}2)$$

EIGENFREQUENCY ANALYSIS

In the eigenfrequency formulation the source terms are absent and you solve for the eigenmodes and the eigenvalues or eigenfrequencies:

$$\nabla \cdot \left(-\frac{1}{\rho_0}\nabla p\right) + \frac{\lambda^2 p}{\rho_0 c_s^2} = 0 \qquad (3\text{-}2)$$

The eigenvalue λ introduced in this equation is related to the eigenfrequency, f, through $\lambda = -i2\pi f$.

Example—Reactive Muffler

Introduction

This model examines the sound-transmission properties of an idealized reactive muffler with infinitely long inlet and outlet pipes (or a reflection-free source at the inlet pipe and a reflection-free end of the outlet pipe) and one expansion chamber. One measure of the transmission properties is the transmission-loss coefficient, D_{tl}, which is defined as

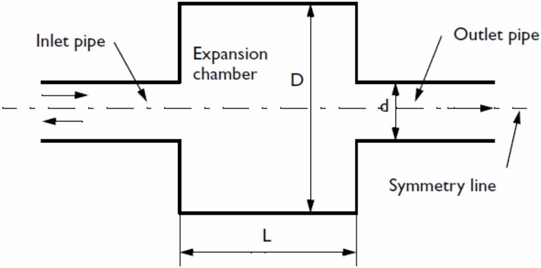
$$D_{\mathrm{tl}} = 10 \cdot \log\left(\frac{W_{\mathrm{i}}}{W_{\mathrm{t}}}\right)$$

where W_{i} is the time-averaged incident sound power and W_{t} is the transmitted sound power. This problem has a theoretical 1D solution that you can compare with the FEM solution.

Model Definition

In the following figure, a plane sound wave enters the inlet pipe (left) and is reflected and attenuated in the expansion chamber. The attenuated sound wave exits through the outlet pipe (right).

The diameter of both the inlet pipe and the outlet pipe is d, and the corresponding cross-sectional area is S_1. The expansion chamber has a diameter D with a corresponding cross-sectional area S_2.



To determine the transmission loss in the model, you must first calculate the incident and transmitted time-averaged sound intensities and the corresponding sound power values. The equation

$$I = \frac{p^2}{2\rho_0 c}$$

gives the time-averaged sound intensities where p is equal to p_0 at the inlet and the computed solution at the outlet.

Using the boundary integration tool, you can evaluate the incident and transmitted sound powers, W, as:

$$W = \int (I \cdot 2\pi r) dr$$

According to Ref. 1, the 1D theoretical solution for the transmission loss to this problem is

$$D_{tl} = 10 \cdot \log\left[1 + \left(\frac{S_1}{2 \cdot S_2} - \frac{S_2}{2 \cdot S_1}\right)^2 \cdot (\sin(kL)^2)\right]$$

where k is the wave number; S_1 and S_2 are the areas of the pipes and expansion chamber; and L gives the length of the expansion chamber.

The model computes the pressure, p, for the fluid in the region defined by the above geometry. This is a time-harmonic problem so you can use the Helmholtz equation defined in the axisymmetric Acoustics application mode:

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = 0$$

where $\omega = 2\pi f$ is the angular frequency, ρ_0 is the fluid density, and c_s is the speed of sound. The $\mathbf{q}$ term is a dipole source with the dimension of force per volume.

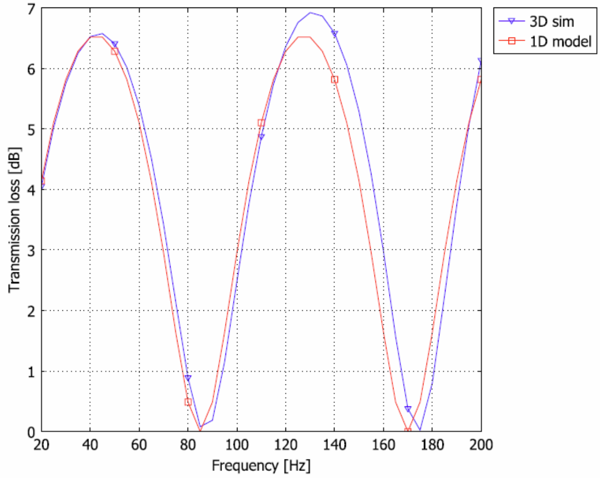


Figure 3-1: Muffler transmission loss versus frequency: theoretical solution (squares) and COMSOL Multiphysics solution (triangles).

Fluid Mechanics

This chapter explains how to use the Incompressible Navier-Stokes application mode for the modeling and simulation of fluid mechanics and fluid statics. Note that the engineering community often uses the term *CFD*, *computational fluid dynamics*, to refer to the numerical simulation of fluids. This chapter concludes with step-by-step instructions on how to model a common benchmark problem: flow over a backward step in the absence of external forces.

The Navier-Stokes Application Mode

Fluid mechanics deals with studies of gases and liquids either in motion (*fluid dynamics*) or at rest (*fluid statics*). When studying liquid flows, it is often safe to assume that the material's density is constant or almost constant. You then have an *incompressible fluid flow*. Using the Incompressible Navier-Stokes application mode you can solve transient and steady-state models of incompressible fluid dynamics.

PDE Formulation and Equations

COMSOL Multiphysics uses a generalized version of the Navier-Stokes equations to allow for variable viscosity.

Starting with the momentum balance in terms of stresses, the generalized equations in terms of transport properties and velocity gradients are

$$\rho\frac{\partial\mathbf{u}}{\partial t} - \nabla\cdot[\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = \mathbf{F} \tag{6-1}$$

$$\nabla\cdot\mathbf{u} = 0$$

The first equation is the *momentum transport equations*, and the second is the *equation of continuity* for incompressible fluids. The following variables and parameters appear in the equations:

- η is the dynamic viscosity
- ρ is the density
- $\mathbf{u}$ is the velocity field
- p is the pressure
- $\mathbf{F}$ is a volume force field such as gravity

Boundary Conditions

The boundary conditions for the Incompressible Navier-Stokes application mode are grouped into the following types:

- Wall
  - No slip (default)
  - Slip
  - Sliding wall
  - Moving/leaking wall
- Inlet
  - Velocity (default)
  - Pressure, no viscous stress
- Outlet
  - Velocity
  - Pressure
  - Pressure, no viscous stress (default)
  - No viscous stress
  - Normal stress

- Symmetry boundary
  - Symmetry (default)
  - Axial symmetry (2D axisymmetry only)
- Open boundary
  - Normal stress (default)
  - No viscous stress
- Stress
  - General stress (default)
  - Normal stress
  - Normal stress, normal flow

Example—Steady Incompressible Flow

Introduction

This model examines the physics of plane, incompressible, and steady flow: flow over a backward step in the absence of external forces. This is a common benchmark problem in CFD. There is no known exact solution, but experimental data has been published (see Ref. 1) making it possible to check the accuracy of the FEM solution. The model includes analyses using both regular triangular meshes and mapped meshes, comparing the solutions for various mesh densities.

Model Definition

Fluid enters from the left side with a parabolic velocity profile, passes over a step, and leaves through the right boundary (Figure 6-5 shows the model geometry).

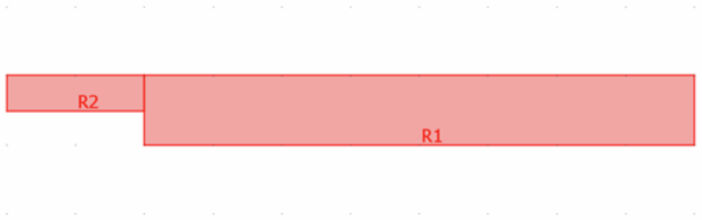


Figure 6-5: The backstep geometry.

The model computes the fluid's velocity components $\mathbf{u} = (u, v)$ in the x and y directions and its pressure p in the region defined by the geometry in the preceding figure. The PDE model for this application uses the stationary incompressible Navier-Stokes equations

$$\begin{cases} -\eta\nabla^2\mathbf{u} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = \mathbf{F} \\ \nabla\cdot\mathbf{u} = 0 \end{cases}$$

- Dynamic viscosity, $\eta = 1.79 \cdot 10^{-5}$ Pa·s
- Density, $\rho = 1.23\ \text{kg/m}^3$
- A force field, $\mathbf{F}$, absent in this model

The first equation is the balance of momentum from Newton's second law. The other relationship is the equation of continuity, where zero on the right-hand side states that the fluid is incompressible.

The shape of the flow pattern depends only on the Reynolds number.

In this model, you choose boundary conditions so that the average velocity at the inlet is $V_{\text{mean}} = 0.544$ m/s. To obtain a corresponding parabolic velocity profile, set $(u, v) = (6V_{\text{mean}}\, s(1-s), 0)$, where s is a boundary parameterization variable that runs from 0 to 1 along the boundary. The fluid is always stationary at the walls, so $(u, v) = (0, 0)$ is the appropriate boundary condition. At the exit boundary, assume a constant static pressure $p = 0$.

For such a fluid flow you can expect a velocity field with a boundary layer of thickness approximately equal to $1/\sqrt{\text{Re}}$ at the walls. To resolve this steep solution gradient you need a few rows of elements across the layer. For a flow with a large Reynolds number, elements in the interior of the channel can be much larger than those near the walls.

USING THE INCOMPRESSIBLE NAVIER-STOKES APPLICATION MODE

Start by setting up a model and solve this problem on a fixed isotropic mesh. The given input data correspond to a Reynolds number of

$$\mathrm{Re} = \frac{0.544 \cdot 2 \cdot 0.0052 \cdot 1.23}{1.79 \cdot 10^{-5}} \approx 389$$

Even though you are working with a steady flow model, it needs initial conditions because the incompressible Navier-Stokes equations are nonlinear. To achieve a numerical solution, the nonlinear solver solves the equations iteratively.

The following plots show examples of the mesh cases:

Case 1: Homogeneous structured mesh

Case 2: Homogeneous unstructured mesh

Case 3: Structured mesh

Case 4: Unstructured mesh

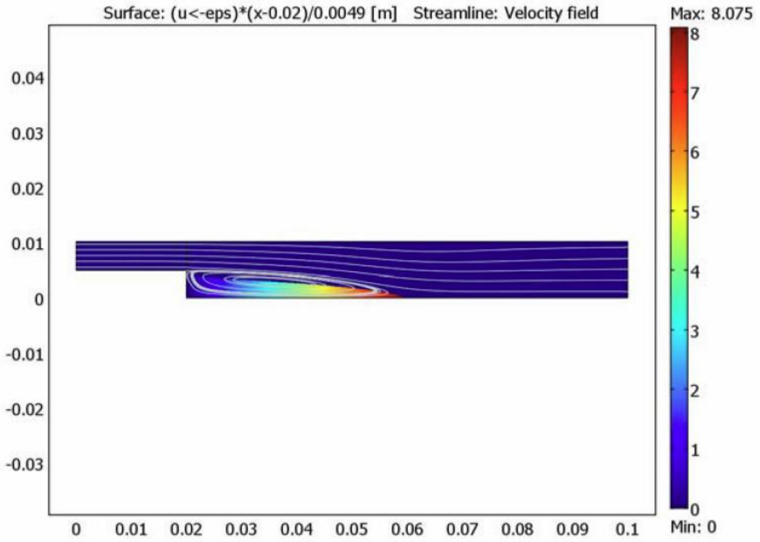
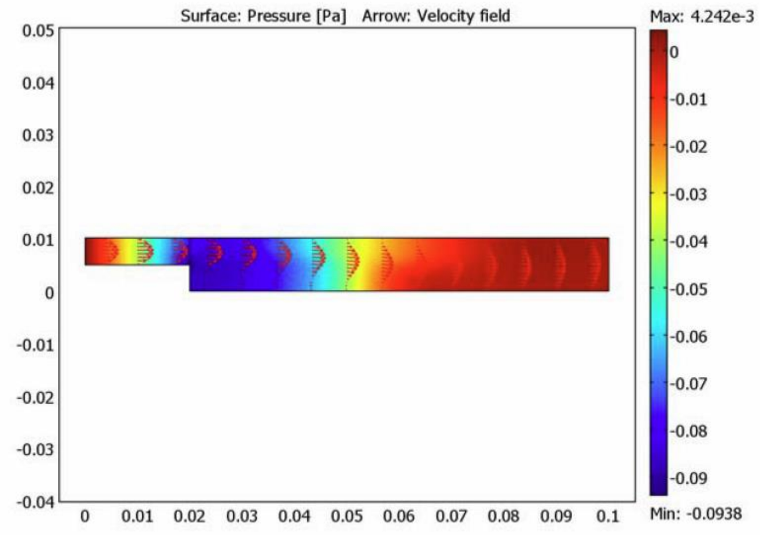


Figure 6-6: A streamline plot of the velocity field.



Surface: Pressure [Pa] Arrow: Velocity field
Max: 4.242e-3
Min: -0.0938
0.05
0.04
0.03
0.02
0.01
0
-0.01
-0.02
-0.03
-0.04
0 0.01 0.02 0.03 0.04 0.05 0.06 0.07 0.08 0.09 0.1
0 -0.01 -0.02 -0.03 -0.04 -0.05 -0.06 -0.07 -0.08 -0.09

Heat Transfer

This chapter covers heat transfer application modes. It starts with some background on heat transfer. It then reviews specifics of the Conduction application mode and then the Convection and Conduction application mode. It concludes with COMSOL Multiphysics models of three heat-transfer examples taken from a NAFEMS (National Agency for Finite Element Methods and Standards) benchmark collection.

Heat Transfer Fundamentals

What Is Heat Transfer?

From the kitchen toaster to the latest high-performance microprocessor, heat is ubiquitous and of great importance in the engineering world. To optimize thermal performance and reduce costs, engineers and researchers are making use of finite element analysis. Because most material properties are temperature-dependent, the effects of heat enter many other disciplines and drive the requirement for multiphysics modeling.

For instance, both the toaster and the microchip contain electrical conductors that generate thermal energy as electric current passes through them. As these conductors release thermal energy, system temperature increases as does that of the conductors. If the electric conductivity is temperature dependent, it changes accordingly and, in turn, affects the electric field in the conductor. Other examples of multiphysics couplings that involve heat transfer are thermal stresses, thermal-fluid convection, and induction heating.

Heat transfer is defined as the movement of energy due to a temperature difference. It is characterized by the following three mechanisms:

- *Conduction* is heat transfer by diffusion in a stationary medium due to a temperature gradient. The medium can be a solid or a fluid.
- *Convective heat transfer* is when heat is transported by a fluid motion. (Engineers sometimes uses convection to refer to heat transfer between either a hot surface and a cold moving fluid or a cold surface and a hot moving fluid.)
- *Radiation* is heat transfer via electromagnetic waves between two surfaces (A and B) with different temperatures T_{A} and T_{B}, provided that Surface A is visible to an infinitesimally small observer on Surface B.

The Heat Equation

The mathematical model for heat transfer by conduction is the *heat equation*:

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q$$

Quickly review the variables and quantities in this equation:

- T is the temperature.
- ρ is the density.
- C_p is the *heat capacity* at constant pressure.
- k is *thermal conductivity.*
- Q is a *heat source* or a *heat sink*.

For a steady-state model, temperature does not change with time, and the first term containing ρ and C vanishes.

If the thermal conductivity is anisotropic, k becomes the thermal conductivity tensor:

$$k = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix}$$

Convection and Conduction application mode as:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \nabla \cdot T = \nabla \cdot (k \nabla T) + Q$$

where **u** is the velocity field. This field can either be provided as a mathematical expression of the independent variables or calculated by a coupling to the velocity field from an application mode such as Incompressible Navier-Stokes.

For transport through conduction and convection, a heat flux vector can be approximated by

$$\mathbf{q} = -k \nabla T + \rho C_p T \mathbf{u}$$

where **q** is the heat flux vector. If the heat transfer is by conduction only, **q** is instead determined by

$$\mathbf{q} = -k \nabla T$$

Note: *Heat capacity* here refers to the quantity that represents the amount of heat required to change one unit of mass of a substance by one degree. It has units of energy per mass per degree (J/(kg·K) in SI units). This quantity is also called *specific heat* or *specific heat capacity.*

Boundary Condition Types

The available boundary conditions are:

BOUNDARY CONDITION	DESCRIPTION
$\mathbf{n}\cdot(k\nabla T) = q_0 + h(T_{\mathrm{inf}} - T) + C_{\mathrm{const}}(T_{\mathrm{amb}}^4 - T^4)$	Heat flux
$\mathbf{n}\cdot(k\nabla T) = 0$	Insulation or symmetry
$T = T_0$	Prescribed temperature
$T = 0$	Zero temperature
$\mathbf{n}_1\cdot(k_1\nabla T_1) = \frac{k}{d}(T_2 - T_1)$ $\mathbf{n}_2\cdot(k_2\nabla T_2) = \frac{k}{d}(T_1 - T_2)$	Thin thermally resistive layer (pair boundaries only)

Examples of Heat Transfer Models

The following heat transfer benchmark examples show how to model heat transfer using:

- Steady-state and transient analysis
- Temperature, heat flux, convective cooling, and radiation boundary conditions
- Thermal conductivity as a function of temperature

1D Steady-State Heat Transfer with Radiation

The first example shows a 1D steady-state thermal analysis including radiation to a prescribed ambient temperature.

Model Definition

This 1D model has a domain of length 0.1 m. The left end is kept at 1000 K, and at the right end there is radiation to 300 K. For the radiation, the model properties are:

- The emissivity, ε, is 0.98.
- The Stefan-Boltzmann constant, σ, is $5.67\cdot10^{-8}$ W/(m^2·K^4).

In the domain, use the following material property:

- The thermal conductivity is 55.563 W/(m·K).

Results

The following plot shows the temperature as a function of position:

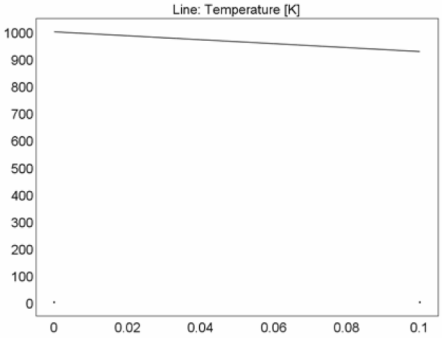


Figure 7-4: Temperature as a function of position.

The benchmark result for the right end is a temperature of 927.0 K. The COMSOL Multiphysics model, using a default mesh with 15 elements, gives a temperature at the end as 926.97 K, which is the exact benchmark value to four significant digits.

2D Steady-State Heat Transfer with Convection

This example shows a 2D steady-state thermal analysis including convection to a prescribed external (ambient) temperature.

Model Definition

This model domain is 0.6 m-by-1.0 m. For the boundary conditions:

- The left boundary is insulated.
- The lower boundary is kept at 100 °C.
- The upper and right boundaries are convecting to 0 °C with a heat transfer coefficient of 750 $W/(m^2 \cdot °C)$.

In the domain use the following material property:

- The thermal conductivity is 52 W/(m·°C).

Results

The following plot shows the temperature as a function of position:

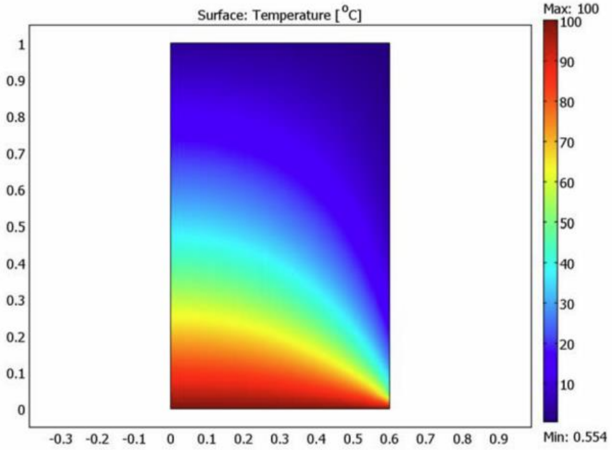


Figure 7-5: Temperature distribution resulting from convection to a prescribed external temperature.

The benchmark result for the target location (x = 0.6 m and y = 0.2 m) is a temperature of 18.25 °C. The COMSOL Multiphysics model, using a default mesh with 556 elements, gives a temperature of 18.28 °C. Successive uniform refinements show temperatures of 18.26 °C and 18.25 °C, converging toward the benchmark result.

2D Axisymmetric Transient Heat Transfer

This example shows an axisymmetric transient thermal analysis with a step change to 1000 °C at time 0.

Model Definition

This model domain is 0.3 m-by-0.4 m. For the boundary conditions, assume the following:

- The left boundary is the symmetry axis.
- The other boundaries have a temperature of 1000 °C. The entire domain is at 0 °C at the start, which represents a step change in temperature at the boundaries.

In the domain use the following material properties:

- The density, ρ, is 7850 kg/m^3
- The heat capacity is 460 J/(kg·°C)
- The thermal conductivity is 52 W/(m·°C)

Results

The following plot shows the temperature as a function of position after 190 seconds:

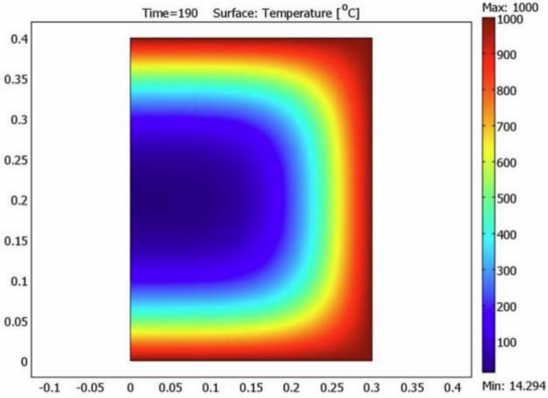


Figure 7-6: Temperature distribution after 190 seconds.

The benchmark result for the target location (r = 0.1 m and z = 0.3 m) is a temperature of 186.5 °C. The COMSOL Multiphysics model, using a default mesh with about 720 elements, gives a temperature of roughly 186.4 °C.

As an additional postprocessing step, map the axisymmetric solution to 3D using an extrusion coupling variable to show the solution for the entire cylinder (see Figure 7-7).

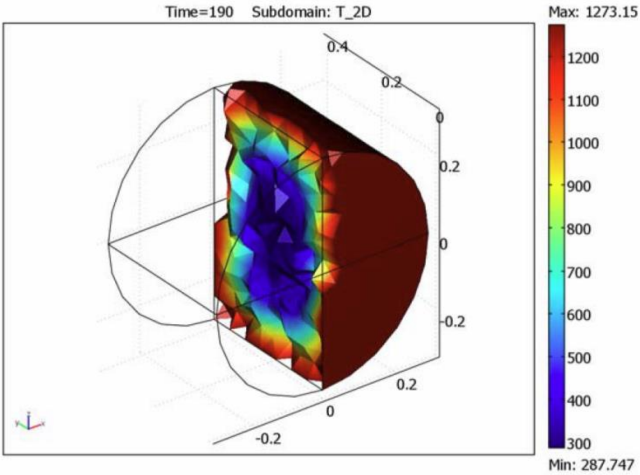


Figure 7-7: Postprocessing of the temperature in the full 3D geometry.

Structural Mechanics

This chapter explains how to use the Structural Mechanics application modes to simulate and analyze applications involving solid mechanics. It begins with a brief theoretical backgrounder on structural mechanics, after which subsequent sections give details of the application modes.

The Structural Mechanics Application Modes

COMSOL Multiphysics includes four application modes for stress analysis and general structural mechanics simulation:

- The Solid, Stress-Strain application mode (for 3D geometries)
- The Plane Stress application mode (for 2D geometries)
- The Plane Strain application mode (for 2D geometries)
- The Axial Symmetry Stress-Strain application mode (for 2D axisymmetric geometries)

The last three cases are 2D simplifications of the full 3D equations, simplifications that are valid under certain assumptions.

Strain-Displacement Relationship

It is possible to completely describe the strain conditions at a point with the deformation components—(u, v, w) in 3D—and their derivatives. You can express the shear strain in a tensor form, ε_{xy}, ε_{yz}, ε_{xz}, or in an engineering form, γ_{xy}, γ_{yz}, γ_{xz}. Following the small-displacement assumption, the normal strain components and the shear strain components are given from the deformation as follows:

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_{xy} = \frac{\gamma_{xy}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \qquad \varepsilon_{yz} = \frac{\gamma_{yz}}{2} = \frac{1}{2}\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

$$\varepsilon_z = \frac{\partial w}{\partial z} \qquad \varepsilon_{xz} = \frac{\gamma_{xz}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)$$

The symmetric strain tensor ε consists of both normal and shear strain components:

$$\varepsilon = \begin{bmatrix} \varepsilon_x & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{xy} & \varepsilon_y & \varepsilon_{yz} \\ \varepsilon_{xz} & \varepsilon_{yz} & \varepsilon_z \end{bmatrix}$$

Stress-Strain Relationship

The stress in a material is described by the symmetric stress tensor

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \qquad \tau_{xy} = \tau_{yx} \qquad \tau_{xz} = \tau_{zx} \qquad \tau_{yz} = \tau_{zy}$$

consisting of three normal stresses (σ_x, σ_y, σ_z) and six, or if symmetry is used, three shear stresses (τ_{xy}, τ_{yz}, τ_{xz}). The stress-strain relationship for linear conditions reads:

$$\sigma = D\varepsilon$$

where D is the 6×6 elasticity matrix, and the stress and strain components are described in vector form with the six stress and strain components in column vectors defined as

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}$$

Note: The following descriptions use the compact notation σ and ε, meaning either the stress/strain vector or tensor depending on the context.

The elasticity matrix D and the more basic matrix D^{-1} (the inverse of D, also known as the flexibility or compliance matrix) are defined differently for isotropic, orthotropic, and anisotropic material. For isotropic materials, the D^{-1} matrix looks like

$$D^{-1} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix}$$

where E is the modulus of elasticity (also known as *Young's modulus*), and ν is *Poisson's ratio*, which defines contraction in the perpendicular direction. Inverting D^{-1}

$$D = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

EQUILIBRIUM EQUATION

The equilibrium equations expressed in the stresses for 3D are

$$-\frac{\partial \sigma_x}{\partial x} - \frac{\partial \tau_{xy}}{\partial y} - \frac{\partial \tau_{xz}}{\partial z} = F_x$$

$$-\frac{\partial \tau_{xy}}{\partial x} - \frac{\partial \sigma_y}{\partial y} - \frac{\partial \tau_{yz}}{\partial z} = F_y$$

$$-\frac{\partial \tau_{xz}}{\partial x} - \frac{\partial \tau_{yz}}{\partial y} - \frac{\partial \sigma_z}{\partial z} = F_z$$

where **F** denotes the volume forces (body forces).

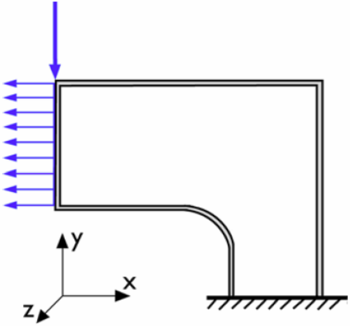
Using compact notation, you can write this relationship as

$$-\nabla \cdot \sigma = \mathbf{F}$$

where σ is the stress tensor. Substituting the stress-strain and strain-displacement relationships in the above equation results in Navier's equation expressed in the displacement.

The Plane Stress Application Mode

Use the Structural Mechanics Plane Stress application mode to analyze thin in-plane loaded plates. This application mode solves for the global displacements (u, v) in the x and y directions. In a state of plane stress the σ_z, τ_{yz}, and τ_{xz} components of the stress tensor are assumed to be zero.

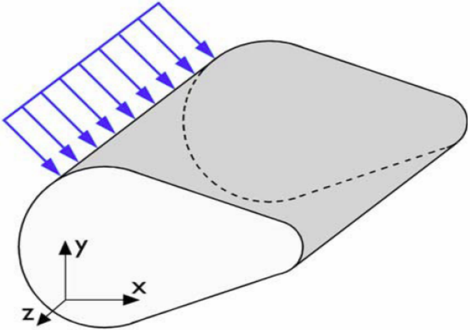


Loads and constraints in a plane stress analysis.

Loads in the x and y directions are allowed. The mode assumes that the loads are constant throughout the thickness of the material but that thickness can vary in the x and y directions. The plane stress condition prevails in a thin flat plate in the xy-plane loaded only in its own plane and without any z-direction restraint.

The Plane Strain Application Mode

The Plane Strain application mode solves for the global displacements (u, v) in the x and y directions. In a state of plane strain, the ε_z, ε_{yz}, and ε_{xz} components of the strain tensor are assumed to be zero.



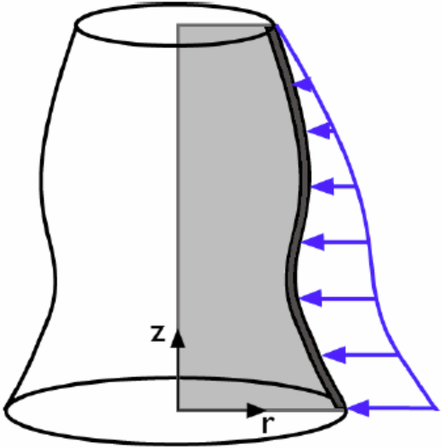
Loads in a plane strain analysis.

Loads in the x and y directions are allowed. The loads are assumed to be constant throughout the thickness of the material, but that thickness can vary in the x and y directions. The plane strain condition prevails in geometries that extend much farther in the z direction than in the x and y directions, or when the z-displacement is in some way restricted. The 2D geometry in a plane strain model represents a cross section that cuts a very long or infinite depth such that you can ignore any end effects. An example is a long tunnel along the z-axis where it is sufficient to study a unit-depth slice in the xy-plane. A plane strain model is sometimes also referred to as a *unit-depth model.*

The Axial Symmetry, Stress-Strain Application Mode

The Axial Symmetry, Stress-Strain application mode uses the cylindrical coordinates r, φ, and z. It solves the equations for the global displacement (u, w) in the r and z directions. The displacement v in the φ direction together with the $\tau_{r\varphi}$, $\tau_{\varphi z}$, $\gamma_{r\varphi}$, and $\gamma_{\varphi z}$ components of the stresses and strains are assumed to be zero. In this mode, loads are independent of φ, and it allows them only in the r and z directions.

You can view the domain where the equations are solved as the intersection between the original axisymmetric 3D solid and the half plane $\varphi = 0$, $r \geq 0$. Therefore it is necessary to draw the geometry only in the half plane $r \geq 0$. Later on, recover the original 3D solid by rotating the 2D geometry about the z-axis (see the figure below).



Loads in an axisymmetric stress-strain analysis. The modeling domain is the gray 2D section.

Displacement Formulation

We now wish to develop the reduced set of field equations solely in terms of the displacements. This system is referred to as the *displacement formulation* and is most useful when combined with displacement-only boundary conditions found in the Problem 2 statement. This development is somewhat more straightforward than our previous discussion for the stress formulation. For this case, we wish to eliminate the strains and stresses from the fundamental system (5.1.5). This is easily accomplished by using the strain-displacement relations in Hooke's law to give

$$\sigma_{ij} = \lambda u_{k,k}\delta_{ij} + \mu(u_{i,j} + u_{j,i}) \tag{5.4.1}$$

which can be expressed as six scalar equations

$$
\begin{aligned}
\sigma_x &= \lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+2\mu\frac{\partial u}{\partial x}\\
\sigma_y &= \lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+2\mu\frac{\partial v}{\partial y}\\
\sigma_z &= \lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+2\mu\frac{\partial w}{\partial z}\\
\tau_{xy} &= \mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right),\ \tau_{yz}=\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right),\ \tau_{zx}=\mu\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)
\end{aligned}
\tag{5.4.2}
$$

Using these relations in the equilibrium equations gives the result

$$
\mu u_{i,\,kk} + (\lambda+\mu)u_{k,\,ki} + F_i = 0 \tag{5.4.3}
$$

which are the equilibrium equations in terms of the displacements and are referred to as *Navier's* or *Lamé's equations*. This system can be expressed in vector form as

$$
\mu\nabla^2\boldsymbol{u} + (\lambda+\mu)\nabla(\nabla\cdot\boldsymbol{u}) + \boldsymbol{F} = 0 \tag{5.4.4}
$$

or written out in terms of the three scalar equations

$$
\begin{aligned}
\mu\nabla^2 u + (\lambda+\mu)\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) + F_x = 0 \\
\mu\nabla^2 v + (\lambda+\mu)\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) + F_y = 0 \\
\mu\nabla^2 w + (\lambda+\mu)\frac{\partial}{\partial z}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) + F_z = 0
\end{aligned}
\tag{5.4.5}
$$

where the Laplacian is given by $\nabla^2 = (\partial^2/\partial x^2) + (\partial^2/\partial y^2) + (\partial^2/\partial z^2)$. Navier's equations are the desired formulation for the displacement problem, and the system represents three equations for the three unknown displacement components. Similar to the stress formulation, this system is still difficult to solve, and additional mathematical techniques have been developed to further simplify these equations for problem solution. Common methods normally employ the use of *displacement potential functions*. It is shown in Chapter 13 that several such schemes can be developed that allow the displacement vector to be expressed in terms of particular potentials. These schemes generally simplify the problem by yielding uncoupled governing equations in terms of the displacement potentials. This then allows several analytical methods to be employed to solve problems of interest. Several of these techniques are discussed in later sections of the text.

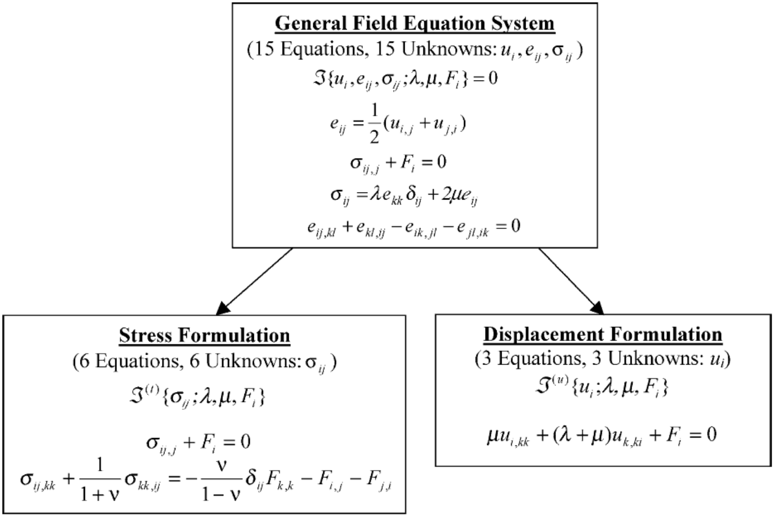


FIGURE 5-5 *Schematic of elasticity field equations.*

FREQUENCY RESPONSE ANALYSIS

Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. Examples of oscillatory excitation include rotating machinery, unbalanced tires, and helicopter blades. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and/or enforced motions (displacements, velocities, or accelerations).

Oscillatory loading is sinusoidal in nature. In its simplest case, this loading is defined as having an amplitude at a specific frequency. The steady-state oscillatory response occurs at the same frequency as the loading. The response may be shifted in time due to damping in the system. The shift in response is called a phase shift because the peak loading and peak response no longer occur at the same time. An example of phase shift is shown in Figure 5-1.

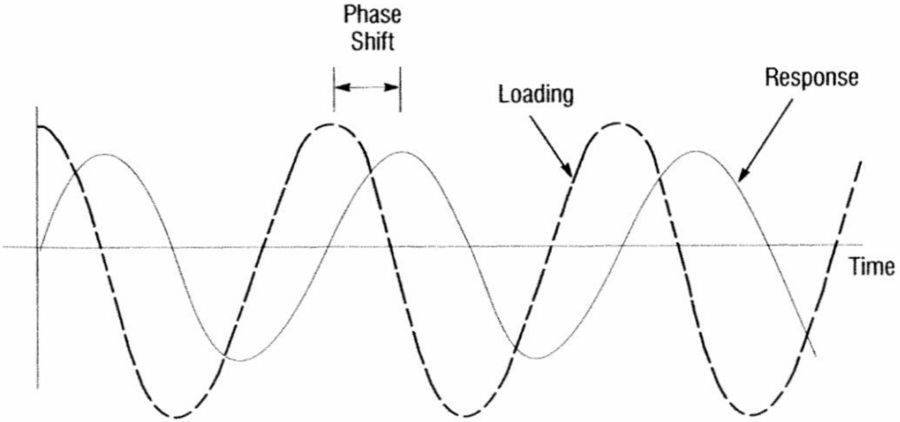


Figure 5-1. Phase Shift.

In direct frequency response analysis, structural response is computed at discrete excitation frequencies by solving a set of coupled matrix equations using complex algebra. Begin with the damped forced vibration equation of motion with harmonic excitation

$$[M]\{\ddot{x}(t)\} + [B]\{\dot{x}(t)\} + [K]\{x(t)\} = \{P(\omega)\}e^{i\omega t} \qquad (5\text{-}1)$$

The load in Eq. (5-1) is introduced as a complex vector, which is convenient for the mathematical solution of the problem. From a physical point of view, the load can be real or imaginary, or both. The same interpretation is used for response quantities.

For harmonic motion (which is the basis of a frequency response analysis), assume a harmonic solution of the form:

$$\{x\} = \{u(\omega)\}e^{i\omega t} \tag{5-2}$$

where $\{u(\omega)\}$ is a complex displacement vector. Taking the first and second derivatives of Eq. (5-2), the following is obtained:

$$\begin{aligned} \{\dot{x}\} &= i\omega\{u(\omega)\}e^{i\omega t} \\ \{\ddot{x}\} &= -\omega^2\{u(\omega)\}e^{i\omega t} \end{aligned} \tag{5-3}$$

When the above expressions are substituted into Eq. (5-1), the following is obtained:

$$-\omega^2[M]\{u(\omega)\}e^{i\omega t} + i\omega[B]\{u(\omega)\}e^{i\omega t} + [K]\{u(\omega)\}e^{i\omega t} = \{P(\omega)\}e^{i\omega t} \tag{5-4}$$

which after dividing by $e^{i\omega t}$ simplifies to

$$\left[-\omega^2 M + i\omega B + K\right]\{u(\omega)\} = \{P(\omega)\} \tag{5-5}$$

The equation of motion is solved by inserting the forcing frequency ω into the equation of motion. This expression represents a system of equations with complex coefficients if damping is included or the applied loads have phase angles. The equations of motion at each input frequency are then solved in a manner similar to a statics problem using complex arithmetic.

Navier's equation of motion

The actual position of a displaced particle is $\boldsymbol{x}' = \boldsymbol{x} + \boldsymbol{u}(\boldsymbol{x}, t)$, and since its original position $\boldsymbol{x}$ is time independent, its actual velocity is $\boldsymbol{v}(\boldsymbol{x}, t) = \partial \boldsymbol{u}(\boldsymbol{x}, t)/\partial t$ and its acceleration $\boldsymbol{w}(\boldsymbol{x}, t) = \partial^2 \boldsymbol{u}(\boldsymbol{x}, t)/\partial t^2$. Newton's Second Law — mass times acceleration equals force — applied to every material particle in the body takes the form, $dM\boldsymbol{w} = \boldsymbol{f}^* dV$. Dividing by dV and re-using the effective force density for an isotropic homogeneous elastic material from the left hand side of the equation of equilibrium (12-2), we arrive at *Navier's equation of motion* (1821),

$$\boxed{\rho \frac{\partial^2 \boldsymbol{u}}{\partial t^2} = \boldsymbol{f} + \mu \boldsymbol{\nabla}^2 \boldsymbol{u} + (\lambda + \mu) \boldsymbol{\nabla} \boldsymbol{\nabla} \cdot \boldsymbol{u} \ .} \tag{14-1}$$

Here λ, μ, and ρ are as before are assumed to be material parameters that do not depend on space and time. In the case that they depend on the spatial position $\boldsymbol{x}$, as they do in Earth's solid mantle, Navier's equation of motion takes a somewhat different form (see problem 14.1). This equation of motion reduces by construction to Navier's equilibrium equation for a time-independent displacement. As in elastostatics, the displacement field and the stress vector must be continuous across material interfaces.

It must be emphasized that Navier's equation of motion is only valid in the limit of small and smooth displacement fields. If the displacement gradients are large, non-linear terms will first of all appear in the strain tensor (10-44), but there will also arise non-linear terms from the derivatives of the stress tensor in the effective force, as demonstrated by eq. (12-3). In chapter 15 we shall derive the correct equations of motion for continuous matter (in the Euler representation) with all such terms included.

Harmonic analysis

A general mathematical theorem due to Fourier tells us that any time-dependent function may be resolved in a superposition of *harmonic* or *monochromatic* components, each oscillating with a single frequency. For linear differential equations — ordinary or partial — with time-independent coefficients this is particularly advantageous because it reduces the time-dependent problem to a time-independent one (for each frequency).

A real harmonic displacement field with *circular frequency* ω and *period* $2\pi/\omega$ satisfies the equation,

$$\frac{\partial^2 \boldsymbol{u}}{\partial t^2} = -\omega^2 \boldsymbol{u} \ . \tag{14-9}$$

The most general solution is a linear superposition of two time-independent *standing wave fields* $\boldsymbol{u}_1(\boldsymbol{x})$ and $\boldsymbol{u}_2(\boldsymbol{x})$,

$$\boldsymbol{u}(\boldsymbol{x}, t) = \boldsymbol{u}_1(\boldsymbol{x}) \cos \omega t + \boldsymbol{u}_2(\boldsymbol{x}) \sin \omega t \ . \tag{14-10}$$

Instead of working with two real fields it is often most convenient to collect them in a single *complex* time-independent standing-wave field,

$$\boldsymbol{u}(\boldsymbol{x}) = \boldsymbol{u}_1(\boldsymbol{x}) + i\boldsymbol{u}_2(\boldsymbol{x}) \ . \tag{14-11}$$

The harmonic displacement field then becomes the real part of a complex field,

$$\boldsymbol{u}(\boldsymbol{x}, t) = \mathcal{R}e \left[\boldsymbol{u}(\boldsymbol{x}) \, e^{-i\omega t} \right] \ . \tag{14-12}$$

The displacement velocity is correspondingly given by the imaginary part,

$$\frac{\partial \boldsymbol{u}(\boldsymbol{x}, t)}{\partial t} = \omega \mathcal{I}m \left[\boldsymbol{u}(\boldsymbol{x}) \, e^{-i\omega t} \right] \ , \tag{14-13}$$

as may easily be verified.

Since the wave equation (14-2) is linear in $\boldsymbol{u}$, it is also satisfied by the velocity field $\partial\boldsymbol{u}/\partial t$ and thus by both the real and imaginary part of the complex field $\boldsymbol{u}(\boldsymbol{x})e^{-i\omega t}$, *i.e.* by the whole complex field itself. Inserting this field into the wave equation we obtain a single time-independent equation for the complex standing-wave field $\boldsymbol{u}(\boldsymbol{x})$,

$$\boxed{-\rho\omega^2\boldsymbol{u} = \mu\boldsymbol{\nabla}^2\boldsymbol{u} + (\lambda+\mu)\boldsymbol{\nabla}\boldsymbol{\nabla}\cdot\boldsymbol{u}\ .} \tag{14-14}$$

It may be viewed as an *eigenvalue equation* for the operator $\mu\delta_{ij}\boldsymbol{\nabla}^2+(\lambda+\mu)\nabla_i\nabla_j$ with eigenfunction $\boldsymbol{u}(\boldsymbol{x})$ and $-\rho\omega^2$ as eigenvalue. It may be shown that ω^2 is always real and positive (problem 14.4). In a finite body, the boundary conditions only allow solutions for a discrete set of eigenfrequencies, whereas in an infinite medium the eigenfrequencies normally form a continuum.

The harmonic analysis may immediately be extended to Navier's equation of motion with a time-dependent body force field $\boldsymbol{f}(\boldsymbol{x},t)$. This will only add the complex harmonic amplitude $\boldsymbol{f}(\boldsymbol{x})$ of the force field to the right hand side of (14-14).

The solution of the equation of motion for natural frequencies and normal modes requires a special reduced form of the equation of motion. If there is no damping and no applied loading, the equation of motion in matrix form reduces to

$$[M]\{\ddot{u}\} + [K]\{u\} = 0 \tag{3-1}$$

where $[M]$ = mass matrix

$[K]$ = stiffness matrix

This is the equation of motion for undamped free vibration. To solve Eq. (3-1) assume a harmonic solution of the form

$$\{u\} = \{\phi\} \sin \omega t \tag{3-2}$$

where $\{\phi\}$ = the eigenvector or mode shape

ω = is the circular natural frequency

Aside from this harmonic form being the key to the numerical solution of the problem, this form also has a physical importance. The harmonic form of the solution means that all the degrees of freedom of the vibrating structure move in a synchronous manner. The structural configuration does not change its basic shape during motion; only its amplitude changes.

If differentiation of the assumed harmonic solution is performed and substituted into the equation of motion, the following is obtained:

$$-\omega^2[M]\{\phi\}\sin\omega t + [K]\{\phi\}\sin\omega t = 0 \tag{3-3}$$

which after simplifying becomes

$$\left([K] - \omega^2[M]\right)\{\phi\} = 0 \tag{3-4}$$

This equation is called the eigenequation, which is a set of homogeneous algebraic equations for the components of the eigenvector and forms the basis for the eigenvalue problem. An eigenvalue problem is a specific equation form that has many applications in linear matrix algebra. The basic form of an eigenvalue problem is

$$[A - \lambda I]x = 0 \tag{3-5}$$

where A = square matrix

λ = eigenvalues

I = identity matrix

x = eigenvector

There are two possible solution forms for Eq. (3-4):

1. If $\det\left([K] - \omega^2[M]\right) \neq 0$, the only possible solution is

$$\{\phi\} = 0 \tag{3-6}$$

This is the trivial solution, which does not provide any valuable information from a physical point of view, since it represents the case of no motion. ("det" denotes the determinant of a matrix.)

2. If det $\left([K] - \omega^2[M]\right) = 0$, then a non-trivial solution $\left(\{\phi\} \neq 0\right)$ is obtained for

$$\left([K] - \omega^2[M]\right)\{\phi\} = 0 \tag{3-7}$$

From a structural engineering point of view, the general mathematical eigenvalue problem reduces to one of solving the equation of the form

$$\det\left([K] - \omega^2[M]\right) = 0 \tag{3-8}$$

or

$$\det\left([K] - \lambda[M]\right) = 0 \tag{3-9}$$

where $\lambda = \omega^2$

The determinant is zero only at a set of discrete eigenvalues λ_i or ω_i^2. There is an eigenvector $\{\phi_i\}$ which satisfies Eq. (3-7) and corresponds to each eigenvalue. Therefore, Eq. (3-7) can be rewritten as

$$\left[K - \omega_i^2 M\right]\{\phi_i\} = 0 \qquad i = 1,\ 2,\ 3... \tag{3-10}$$

Each eigenvalue and eigenvector define a free vibration mode of the structure. The i-th eigenvalue λ_i is related to the i-th natural frequency as follows:

$$f_i = \frac{\omega_i}{2\pi} \tag{3-11}$$

where f_i = i-th natural frequency

$\omega_i = \sqrt{\lambda_i}$

The number of eigenvalues and eigenvectors is equal to the number of degrees of freedom that have mass or the number of dynamic degrees of freedom.

There are a number of characteristics of natural frequencies and mode shapes that make them useful in various dynamic analyses. First, when a linear elastic structure is vibrating in free or forced vibration, its deflected shape at any given time is a linear combination of all of its normal modes

$$\{u\} = \sum_i \{\phi_i\} \xi_i \tag{3-12}$$

where $\{u\}$ = vector of physical displacements

$\{\phi_i\}$ = i-th mode shape

ξ_i = i-th modal displacement