COSMOS/M

A COMPLETE FINITE ELEMENT ANALYSIS SYSTEM

2.7

Basic System Finite Element Analysis
Part 1

STRUCTURAL RESEARCH & ANALYSIS CORP.

First Edition
COSMOS/M 2.7
December 2001

Copyright

Disclaimer

The authors have taken due care in preparing this manual and the examples presented herein. In no event shall SRAC assume any liability or responsibility to any person or company for direct or indirect damage resulting from the use of the information contained herein or any discrepancies between this information and the actual operation of the software.

Licenses & Trademarks





Contents







1 About the Basic FEA System

Introduction

COSMOS/M a complete, modular, self-contained finite element system developed by Structural Research and Analysis Corporation for personal computers and workstations. The program includes modules to solve linear and nonlinear static and dynamic structural problems, in addition to solving problems in the fields of structural optimization, heat transfer, fluid mechanics and electromagnetics. Modules for special problems like fatigue are also available. The system is constantly developed and maintained by using state-of-the-art techniques along with up-to-date hardware capabilities. It is designed to satisfy its users needs. The modularity of the system facilitates its upgrading as the needs of the user grow.

The Basic FEA System is composed of three modules: GEOSTAR for model creation and results display, STAR for linear static analysis, and DSTAR for buckling and modal analysis. The integrated system lets you model, analyze, and evaluate your design within one graphical environment.

This chapter focuses on presenting some of the salient features of finite element modeling and analysis you can apply using the Basic System. The mathematical background for some of the analysis features presented here are explained in the next chapter. To preserve clarity in presentation, and moreover owing to space limitations, the documentation for GEOSTAR is provided to you *separately* in two parts: COSMOS/M User Guide (V. 1), and COSMOS/M Command Manual (V. 2).

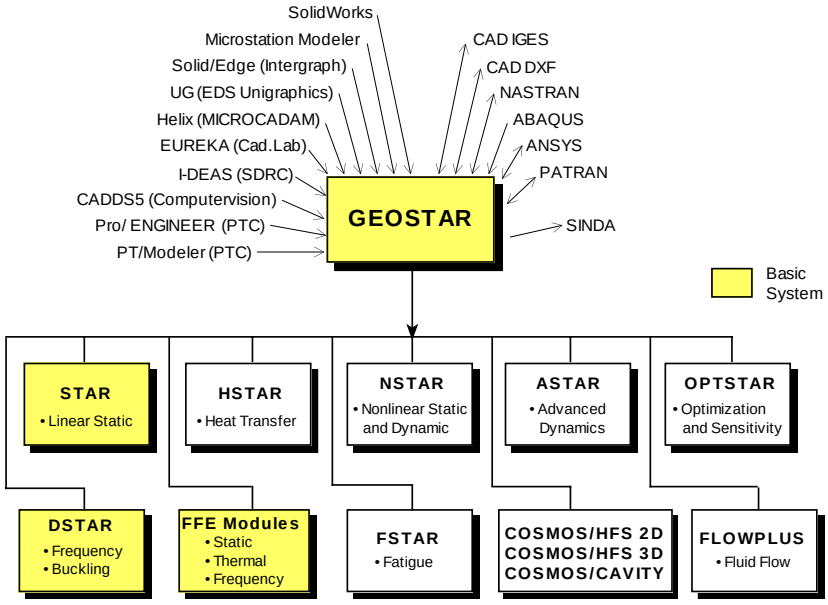



✍ The ***Basic FEA System*** or ***Basic System*** quoted throughout this manual therefore refers to linear static, natural frequency, and buckling analysis only.

COSMOS/M Structure

The COSMOS/M system consists of a pre- and postprocessor, various analysis modules, interfaces, translators and utilities as shown in Figure 1-1. The Basic System modules, highlighted in the figure, are the ones that are most commonly used for the evaluation of mechanical systems including linear static, buckling, and natural frequency and mode shape analyses.

Figure 1-1. COSMOS/M Modules and Translators



GEOSTAR module is the pre- and postprocessor of the COSMOS/M finite element system. It is the *nucleus* of the COSMOS/M system, providing the graphical user interface (GUI) for geometric modeling and finite element analysis. In a typical COSMOS/M session, you will first enter GEOSTAR, build your model using various geometry and meshing tools, apply loads and boundary conditions, and

then subject it to analysis. At this point, you will automatically exit GEOSTAR and link up with the required COSMOS/M analysis module. After completion of analysis, the program automatically returns to the GEOSTAR graphical environment for postprocessing.

GEOSTAR is an interactive full three-dimensional CAD-like graphic geometric modeler, mesh generator and a powerful tool for FEA pre- and postprocessing. The user can create the model, provide all related analysis information, choose the type of analysis, review, plot and print the results, without leaving the GEOSTAR screen. The program is powerful, intuitive and easy to learn. The average user can quickly learn to create and solve real life problems after a few hours of training.

The GEOSTAR program, as shown in Figure 1-1, controls the analysis modules and provides an interaction environment among them. These modules are as shown below in Table 1-1:

Table 1-1. Functions of COSMOS/M Modules

Basic System	STAR DSTAR	Linear Static Analysis Buckling, Frequency and Mode Shape Analysis
Advanced Modules	ASTAR ESTAR FSTAR HSTAR NSTAR OPTSTAR HFS Suite FLOWPLUS FLOWSTAR	Advanced Dynamic Linear Analysis Electromagnetic Analysis Fatigue Analysis Heat Transfer Analysis Nonlinear Static and Dynamic Analysis Structural Optimization High Frequency Electromagnetic Analysis Turbulent Fluid Flow Analysis Fluid Flow Analysis (discontinued)

In addition, GEOSTAR can transfer data and information between COSMOS/M and other CAD and FEA packages, as shown in Figure 1-1. These programs are:

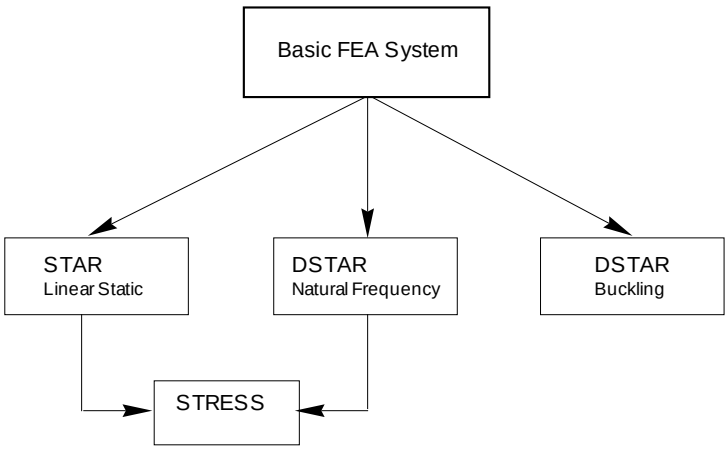
Table 1-2. COSMOS/M CAD and FEA Translators

CAD	IGES DXF
FEA	ABAQUS ANSYS NASTRAN PATRAN SINDA TEAP

Basic System Modules

The Basic System consists of two main modules STAR and DSTAR, and a sub-module, STRESS, which performs stress calculations. These modules interact with each other as shown in the figure below:

Figure 1-2. Basic System Modules



The linear static analysis module (STAR) uses the linear theory of structures, based on the assumption of small displacements, to calculate structural deformations. For many structures (for example, frames), the computation of stresses is often unnecessary. To provide this flexibility, the computation of stresses is performed in a separate submodule called STRESS in COSMOS/M. The module STAR calls the STRESS submodule to calculate the element and nodal stresses for most elements. The DSTAR module evaluates natural frequencies and the corresponding mode shapes of a system. The module can *also* calculate the buckling loads and the associated mode shapes. For modal analysis, the STRESS submodule also calculates the element and nodal *virtual* stresses for most elements based on the results from DSTAR.

The Linear Static Analysis Module (STAR)

STAR uses the linear theory of structures, based on the assumption of small displacements, to calculate structural deformations. As mentioned earlier, STAR calls the STRESS submodule to calculate stresses. The STRESS submodule

calculates element and nodal stresses for most elements based on the results from STAR or DSTAR (modal analysis only). Stresses for multiple load cases are obtained in a single run and the combination of load cases is possible in the post-processing stage. Stresses can be obtained in any defined coordinate systems. The STRESS module supports all of STAR features.

In a typical linear static stress analysis, you will determine the stresses, displacements, strains, and reactions in the finite element model. The analysis is linear if the nonlinearities due to various sources can be either *linearized* or completely ignored. Results from a linear analysis include nodal displacements, nodal and element stresses, forces, reactions, etc. These results can be graphically viewed on the screen or inspected in the output file.

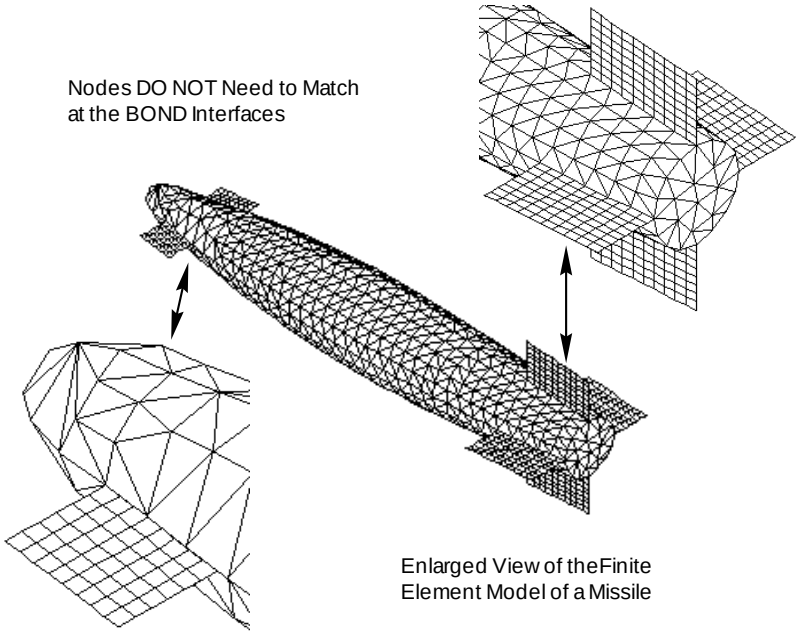
The following are some important features of the linear static stress analysis module STAR:

- Extensive element library (see Chapter 5, *Element Library*).
- Isotropic, orthotropic, anisotropic and composite material properties.
- Temperature-dependent material properties.
- Failure criteria for composite elements.
- Prescribed displacements, with or without other loadings.
- Coupled degrees of freedom and constraint equations.
- Thermal, gravitational and centrifugal loads.
- Beam loading.
- In-plane effects in the stiffness evaluation (geometric or differential stiffness).
- Multiple load cases in a single run.
- Soft spring option to prevent instabilities.
- Substructuring capability (for large problems) to build and analyze the chosen superelements through condensation and recovery process.
- Fluid-solid interaction.
- Gap-friction problems.
- Grid-force balance and reaction force calculation.
- Asymmetric loading of axisymmetric models.
- Strain energy and error calculations.




- 2D and 3D crack element options: the program computes the stress intensity factors for all three modes of fracture.
- Adaptive P-, H- and H-P versions of the adaptive finite element method, including three dimensional problems using tetrahedral elements.
- Bonding of (or connecting) non-compatible parts at the common boundary of the same model even if nodes and element types do not match. Solid-solid, solid-shell and shell-shell bonds can be made by specifying the interface geometric entity. This feature can be used to make curve-to-curve, curve-to-surface and surface-to-surface bonding of your models with non-compatible meshes; see Chapter 8, *Modeling Guidelines*, for more information. The figure below shows an example where the bond capability can be applied. This model consists of tetrahedron solid elements in the hull of the missile to which the fins modeled with shell elements are connected.

Figure 1-3. Bonding of Non-Compatible Element Meshes



- Combining Static and Dynamic Analysis Results: you can link the results of an advanced dynamic analysis (performed in the ASTAR module) at any specified time step or frequency with static analysis results during postprocessing; see Chapter 10, *Postprocessing*, for more information.
- Coupling of Analysis Results: you can link the results of other COSMOS/M analyses, such as heat transfer, fluid flow, and electromagnetics, to include their effects as loads in the stress and strain calculations performed in the Basic System. This coupling is *unidirectional* (see Chapter 12, *Multidisciplinary Analysis)* for performing interaction analysis, as indicated in the following figure.
- Submodeling:

  After you run your problem with a relatively coarse mesh, you may define the areas of concern as submodels where stresses may not have been calculated accurately due to sharp corners, geometric non-uniformity, and/or load concentration. With submodeling, you need only to solve for the areas of concern. Submodeling accelerates the job of obtaining accurate results in areas of stress concentration.
- Multi-Thermal load cases:

  a. User-defined nodal temperature profiles may be defined for every primary load cases.

  b. Transient thermal stress analysis where temperature profiles are read directly from the results of thermal analysis. The temperature profile from a particular time step may be assigned to a primary load case. Results are then calculated for all load cases as usual.
- ASME stress check:

  An evaluation of stress results of FEA solution in a form suitable for comparison to the ASME code section III stress requirements is formulated. Formulations are developed to compute the membrane, bending, peak stress intensities on arbitrary cross-sectional slices.
- Saving of the decomposed stiffness matrix:

  The decomposed stiffness matrix from static, frequency or buckling analysis is saved for subsequent run by any other analysis to save time and effort by reading the decomposed matrix rather than recalculating it.
- Inertia Relief:

  Balancing the external load on the accelerating bodies such as rockets by their inertia forces.




- Automatic Rigid or Hinge connection:

  Rigid or Hinge connections at the interface of incompatible solids and shell elements with mesh continuity at the interface.

- Support of ADAMS software:

  A direct interface is established between STAR and ADAMS, a product of Mechanical Dynamic Inc., is a leading kinematics and dynamics program. The interface enables you to model real working conditions of your designs by including flexural and kinemetic effects. Loading conditions from ADAMS may be exported to COSMOS/M.

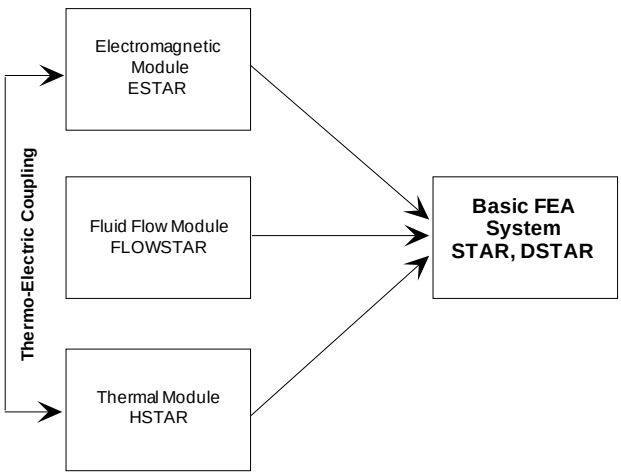
- Application of the J-integral:

  You may use the J-integral in addition to the crack element to compute the stress intensity factors. Available both for rectangular (PLANE2D) and triangular (TRIANG) elements.

- Large File managements:

  You may direct some of the major files of the database to other available drives in order to avoid running out of disk space. You may also instruct the program to break the large files into several segments and write each segment to a different drive.

Figure 1-4. Interaction of Other Analysis Modules with the Basic System



Frequency and Buckling Analysis Module (DSTAR)

The DSTAR module evaluates natural frequencies and the corresponding mode shapes of a system (modal analysis). The module can *also* calculate the buckling loads and the associated mode shapes of eigenvalue buckling problems.

In a typical buckling analysis, the quantities to be computed include the critical loads at which the structure becomes unstable, and the corresponding buckling mode shapes. For eigenvalue buckling, the first few modes are of practical importance. The buckling modes can be plotted or animated during postprocessing.

Modal analysis which determines the natural frequencies and mode shapes is an important phase in the design of many structural components. Similar to buckling, modal analysis involves the computation of eigenvalues, and DSTAR provides many types of eigenvalues extraction techniques. The vibration modes can be plotted or animated on the screen.

The following are some important features of the DSTAR module:

- A variety of eigenvalue extraction procedures:
  - Subspace iteration,
  - Lanczos,
  - Jacobi,
  - Inverse power iteration (one pair only),
  - Guyan Reduction.
- Frequency shift to calculate eigenvalues in a specified range or to treat models with rigid body modes.
- Sturm sequence to check for missed modes.
- Lumped and consistent mass matrices for representing structural mass.
- In-plane effects on stiffness.
- Soft-spring option to treat models with rigid body modes.
- Modal analysis of Piezoelectric Materials: calculates natural frequencies and mode shapes using Hexahedron SOLID elements to account for coupling between elastic and electric fields of piezoelectric materials.
- Non-axisymmetric mode extraction for axisymmetric models.




- Guyan Reduction:

  During design, when changes are being made, Guyan reduction would save significant time. The reduction enables the analyst to exercise some degree of control over the extraction process, selectively ignoring those modes which are of no or little value to a specific analysis of large complex models.

- Automatic Rigid or Hinge connection:

  Rigid or Hinge connections at the interface of incompatible solids and shell elements with mesh continuity at the interface.

- Large File management:

  Partitioning of Large files (such as Stiffness Matrix file) into several drives/ directories.

- Spin Stiffening:

  Accounting for the large displacements effect for spinning structures.

- Interface with the Nonlinear Module (NSTAR) for Frequency and Buckling analyses (Refer to the Advanced Modules manual).

- Bonding of noncompatible meshes (similar to the one described for Static Analysis).

Basic FEA System Limits

The pre- and postprocessing module GEOSTAR and the analysis modules STAR and DSTAR which constitute the Basic System, have the following size limits. The tables below list these limits for the *full version* only.




Table 1-3. Modeling Size Limits Modeling in GEOSTAR

Quantity Preprocessing	Size Limit	Quantity Miscellaneous	Size Limit
Nodes	256,000*	Consecutive zooms	10
Elements	256,000*	Selection sets per entity	10
Keypoints	24,000	Number of saved views	10
Curves	24,000	Text messages	100
Surfaces	8,000	Function keys (GEOFUN file)	50
Volumes	2,000	Number of windows	4
Contours	5,000	Colors	256
Regions	5,000	Length of file (including path)	40
Polyhedra	100	Length of a parameter name (single variable, array, function)	10
Parts	100	No. of parametric arrays	20
Coordinate systems	5,000	No. of user-defined parametric functions:	200
Curves for contour	500	- Length of a function argument name	10
Contours for a region	120	- Number of arguments for a function	20
Surfaces/Regions for a polyhedron	1000	- Length of a function parametric expr.	200
Polyhedra for a part	60	Length of a macro name	10
Bond target surfaces	20	No. of arguments for a macro	10
Bond sets	100	No. of local user defined parametric variables in a macro	50
Material property sets	999	No. of local user defined functions in a macro	50
Real constant sets	5,000	No. of local arrays in a macro	10
Element groups	5,000	No. of command lines starting with # in the entire session file or macro	500

Quantity Postprocessing	Size Limit
Isoplanes	12
Section planes	12
Number of XY curves per plot	6

* GEOSTAR versions supporting 64,000, 128,000, and 256,000 nodes/ elements are automatically installed.




Table 1-4. Analysis: Limits in the Basic FEA System

Quantity	Size Limit
Degrees of freedom	No Limit
Primary load cases	50
Secondary load cases	50
Coupled degrees of freedom	1,500
Constraint equations	1,500
Temperature curves - Points for a temperature curve	100 5,000
Superelements - Super nodes for a superelement - Degrees of freedom associated with a superelement	99 180 2,000
Prescribed non-zero displacements	3,000
Concentrated forces for beam loading	5
Distributed loads for beam loading	3
Number of eigenpairs	200




2 Mathematical Background

Introduction

The previous chapter presented some of the important analysis and modeling features of the Basic System. This chapter presents the mathematical background on some of the important features. COSMOS/M has been *specially* designed to solve finite element problems with a large number of degrees of freedom. The needs of a finite element analyst have been considered in all aspects of design. However, in view of the wide range of possible applications of the program, a high degree of generality and flexibility have been incorporated in many areas of the program. The information provided in this chapter may not discuss all aspects of application of the Basic System.

This chapter only highlights some of the theoretical concepts applicable in the Basic System. For a detailed discussion of the finite element theory, please refer to the Theoretical Manual.

Mathematical Background

The types of analysis you can perform using the Basic FEA System include:

- Linear static analysis
- Eigenvalue buckling analysis
- Natural frequency and mode shape analysis

In the following paragraphs, brief mathematical background for the above types of analysis are presented.

Linear Static Analysis

Static analysis deals with the computation of displacements and stresses due to *static loads*. The term *static loads* refers to loading that does not cause inertial or damping effects to be significant for consideration in the analysis. Static analysis is *linear* if nonlinearities due to plasticity, large deflection, large strain, in-plane effects, contact surfaces, creep and relaxation effects, and other sources can be either *linearized* or completely *ignored*. The procedure for performing a linear static analysis with modeling hints and guidelines is provided in Chapter 8, *Modeling Guidelines* and Chapter 4, *Input Data Requirements*.

The formulation of a linear static problem for solution by the displacement method is fully described by the matrix equation:

$$[K]\{U\} = \{F\} = \{F^a\} + \{F^c\} \quad \textbf{(2-1)}$$

where [K] is the structural (assembled) stiffness matrix, {U} is the vector of unknown nodal displacements, and {F} is the load vector. The load vector {F} has components from mechanical, thermal, and gravitational loads (see Chapter 7, *Loads and Boundary Conditions*, for information). The load vector {F} can be expressed as a combination of applied nodal loads $\{F^a\}$ and reaction (or single point constraint) forces $\{F^c\}$. For linear static problems, each of these load vectors are the superposition of mechanical, thermal, and gravitational loads as shown below:

$$\{F^a\} = \{F^m\} + \{F^{th}\} + \{F^{gr}\} \quad \textbf{(2-2)}$$

The mechanical load vector $\{F^m\}$ is computed as the sum of applied nodal forces and moments, and element pressures as shown below:

$$\left\{F^m\right\} = \left\{F^{nd}\right\} + \sum_{e=1}^{nel} \left\{F_e^{pr}\right\} \quad \textbf{(2-3)}$$

where $\{F^{nd}\}$ is applied nodal load vector, and $\{F_e^{pr}\}$ is the element pressure load vector. The thermal, and gravitational load vectors are computed as follows:

$$\{F^{th}\}=\{F^{nt}\}+\sum_{e=1}^{nel}\{F_e^{th}\} \qquad \{F^{gr}\}=\left(\sum_{e=1}^{nel}[M_e]\right)\{a\} \tag{2-4}$$

where $\{F^{nt}\}$ is the load vector of nodal temperatures, $\{F_e^{th}\}$ is the element thermal load vector, $[M_e]$ is the element mass matrix, and $\{a\}$ is the acceleration vector.

If constraint equations (also known as *multipoint constraints*) which define relationships in displacements between nodes are used, equation (2-1) is partitioned and solved to eliminate some unknowns as explained in the next section. For specified boundary conditions (single point constraints) and prescribed displacements, the structural stiffness matrix is partitioned as shown below:

$$\begin{bmatrix}[K_{ff}] & [K_{fs}]\\ [K_{sf}] & [K_{ss}]\end{bmatrix}\begin{Bmatrix}\{U_f\}\\ \{U_s\}\end{Bmatrix}=\begin{Bmatrix}\{F_f^a\}\\ \{F_s^a\}\end{Bmatrix}+\begin{Bmatrix}\{0\}\\ \{F_s^c\}\end{Bmatrix} \tag{2-5}$$

where the subscript *f* refers to *free* dof's, and s refers to dof's with *specified* values. Equation (2-5) therefore describes the behavior of the structure under forces and fixed and/or prescribed motions.

Constraint Equations (Multipoint Constraints)

Constraint or coupling equations which are also known as multipoint constraints, specify a linear relationship of displacements between two or more nodes. The constraint equations provide one approach to modeling dependent dof's which are defined as linear combinations of one or more other dof's. The other approach to modeling dependent dof's involves the use of rigid links (RBAR element, for example). In COSMOS/M, only the selected components of dof's are considered in the constraint equation, and the remaining components of dof's at these nodes where the constraint equations are applied remain unaltered.

In COSMOS/M, a constraint equation is expressed as a homogeneous linear equation for the dependent dof's in terms of independent dof's as follows:

$$u_i = a_1u_1 + a_2u_2 + a_3u_3 + \ldots + a_nu_n \tag{2-6}$$

where u_i represents the dof's to be constrained (dependent dof's), $u_1 \ldots u_n$ represent the other independent (or retained) dof's, and $a_1 \ldots a_n$ represent the constraint coefficients provided by the user. Equation (2-6) can be symbolically represented as:

$$\{U_m\} = [C_m]\{U_n\} \quad \textbf{(2-7)}$$

where $\{U_m\}$ is the vector of dependent dof's, $\{U_n\}$ is independent dof's, and $[C_m]$ is the constraint equation matrix. The above equations are applied in equation (2-1) after partitioning both sides of equation (2-1) for dependent and independent dof's. In addition, equilibrium conditions require the constraint forces to be added to equation (2-1) before the elimination of $\{U_m\}$.

The benefits of using coupling equations are many; see Chapter 8, **Modeling Guidelines**, and Chapter 11, **Analysis Examples**, for more information.

In-plane Effects in Static Analysis - Stress Stiffening

For many slender structures with in-plane loading, the membrane forces alter the bending stiffness. Buckling occurs when the compressive membrane forces reduce the bending stiffness to zero for a *kinematically* admissible deformation mode. However, when the membrane forces are reversed (tensile instead of compressive), then the bending stiffness is effectively increased. Thus, the structure becomes more *stiff* under tensile in-plane loads, exhibiting what is known as the stress *stiffening* effect. The stress stiffening or softening effect can also be considered in the computation of natural frequencies; see the section, *In-plane Effects in Modal Analysis*, for more information. Buckling is discussed separately in another section.

For these types of structures, the stiffness properties are a function of both the static loads and the deformed shape. While an accurate solution of such problems requires the use of geometrically nonlinear solution techniques, reasonable accuracy can be obtained by using the differential stiffness approach. In this method, it is assumed that the geometrically nonlinear problems can be approximated by adding a *geometric* stiffness matrix K_G (also known as stress, initial stress or differential stiffness matrix, or stability coefficient matrix) to the conventional structural stiffness matrix. The displacements are computed with respect to the original configuration of the structure, and the change in geometry is reflected only in the geometric stiffness matrix. It is also assumed that the magnitude and direction of the loads remain fixed during motion of the structure, and their points of application move with the structure.

Since the geometric stiffness matrix depends on the displacements, the linear static analysis is performed in two stages. In the first stage, the displacements $\{u_i\}$ are computed using the conventional stiffness matrix [K]. In the second stage, the geometric stiffness matrix $[K_G(u_i)]$ is established based on the

computed displacements, $\{u_i\}$, and added to the conventional stiffness matrix [K] to solve for the new displacements, $\{u_{i+1}\}$. The finite element system of equations for linear static stress analysis in the presence of in-plane effects can therefore be written as:

$$([K] + [K_G(u_i)])\{u_{i+1}\} = \{F\} \quad \textbf{(2-8)}$$

Ideally, the displacements $\{u_{i+1}\}$ could be used to compute the new geometric stiffness matrix $[K_G(u_{i+1})]$ and hence compute yet another set of solutions, $\{u_{i+2}\}$, and so on. The iterations can be carried out till successive solutions do not differ by more than the specified tolerance. However, if the applied in-plane (compressive) load is in the vicinity of the buckling load, the iterations may diverge, indicating instability. Such problems warrant the use of buckling analysis, explained in another section. In COSMOS/M, the in-plane effects are considered by performing one iteration only. More options for these iterations are provided if you are using the nonlinear analysis module, NSTAR.

The geometric stiffness matrix in COSMOS/M is *consistent*, built from the same shape functions used to form the conventional stiffness matrix. It is symmetric, but unlike the conventional stiffness matrix, it does not contain terms with elastic moduli. It depends on the element geometry, displacement field, and the state of stress. The geometric stiffness matrix K_G is in general indefinite, and hence cannot be inverted. When used in buckling analysis, the overall structural stiffness matrix which is composed of the normal and geometric stiffness matrix becomes singular with respect to buckling modes.

The in-plane effects in linear static stress analysis can be considered for only one load case.

Treatment of Rigid Body Modes in Static Analysis - Soft Spring Addition

A rigid (free) body mode of a structure represents motion without internal deformations. In structural analysis, rigid body modes arise due to either insufficient or nonexistent constraints imposed on the structure. However, there are many analysis problems where the imposition of constraints is not desirable, yet the deformable behavior of the structure under loads is of interest. For example, you may come across a stress analysis problem where an equilibrium state is maintained by the applied loads, thus requiring no constraints. Ordinarily, for such problems the program will execute unsuccessfully, indicating a message about singular

stiffness matrix. However, if you specify the option for soft spring addition, the program will automatically add a small stiffness to the diagonal terms of the structural stiffness matrix as shown below, enabling numerical stability during solution:

$$\begin{bmatrix} k_{11}+k_{ss} & k_{12} & 0 & k_{14} & 0 & 0 & 0 & \dots \\ & k_{22}+k_{ss} & k_{23} & 0 & 0 & 0 & 0 & \dots \\ & & k_{33}+k_{ss} & k_{34} & 0 & k_{36} & 0 & \dots \\ & & & k_{44}+k_{ss} & k_{45} & k_{46} & 0 & .. \\ & & & & k_{55}+k_{ss} & k_{56} & \dots & \dots \\ & & & & & k_{66}+k_{ss} & \dots & \dots \\ & & & & & & \dots+k_{ss} & \dots \\ & & & & & & & k_{nn}+k_{ss} \end{bmatrix} \quad \text{(2-9)}$$

where k_{ss} denotes the stiffness added due to the soft spring option. The soft spring option is available for all types of analysis in the Basic FEA System, and is specified using the analysis commands (for example, **A_STATIC** (Analysis > STATIC > **Static Analysis Options**), etc.); see Chapter 9, *Performing Analysis*, for more information.

Singular stiffness matrices are also encountered due to ill-posed problems, i.e., models with very low or highly disproportionate elastic moduli and section properties. Owing to the solution method implemented, the use of soft springs is sometimes *required* when you use the gap (contact) elements in the Basic FEA System.

The treatment of rigid body modes in the analysis of natural frequencies can be accomplished in two ways: by using the soft spring option outlined above, and also by using the shift option in eigenvalue extraction. These methods are explained later in the section, *Modal Analysis*.

Substructuring

For the solution of very large finite element models, COSMOS/M provides a procedure known as *substructuring* in which the large structure is divided into smaller parts to be processed in separate executions. These executions generate data for a final merging operation where the solution for the combined structure is obtained. Following this, detailed solutions for each structure can be recovered in separate executions.

In COSMOS/M, each substructure is known as a *superelement*. Each superelement is a complete disjointed structure and it can be connected to a *main* supported structure and/or other superelements. There are limits imposed on the number of superelements and their boundary nodes. It is possible to combine several superelements to form a *pseudo* superelement in what is known as *multilevel* substructuring.

The static stress analysis of a model with substructures is performed in three stages. In the first stage, each superelement is analyzed to produce a description of its behavior as seen at the boundary nodes. In the second stage, the matrices produced for each superelement are combined with the main structure and analyzed. In the third stage, the analysis of individual superelements is completed for displacement and stress recovery at all nodes. The following lines describe these operations in brief.

Consider a superelement for which the finite element system of equations can be written as:

$$[K]\{U\} = \{F\} \qquad \textbf{(2-10)}$$

The substructuring solution process involves partitioning the above equation to obtain partial solution. The partial solution is then merged into the total solution. For the superelement, with superscript I denoting the interior and B denoting the boundary, the above equation can be partitioned as:

$$\begin{bmatrix} K_{II} & K_{IB} \\ K_{BI} & K_{BB} \end{bmatrix} \begin{Bmatrix} U_I \\ U_B \end{Bmatrix} = \begin{Bmatrix} F_I \\ F_B \end{Bmatrix} \qquad \textbf{(2-11)}$$

where U_I represents the displacements at interior nodes, and U_B represents the displacements at boundary or superelement nodes. The above equation can be expanded to:

$$\begin{aligned} [K_{II}]\{U_I\} + [K_{IB}]\{U_B\} &= \{F_I\} \\ [K_{BI}]\{U_I\} + [K_{BB}]\{U_B\} &= \{F_B\} \end{aligned} \qquad \textbf{(2-12)}$$

Solving for the interior displacements leads to:

$$\{U_I\} = [K_{II}]^{-1}\{F_I\} - [K_{II}]^{-1}[K_{IB}]\{U_B\} \qquad \textbf{(2-13)}$$

By inspection, the first term on the right hand side of the above equation is the solution that results from constraining the boundary degrees of freedom and applying the external loads on the interior nodes. It is therefore a partial solution

of the interior nodes. The remaining solution component at the interior nodes is given by applying the second part of the above right hand side in equation (2-12) which yields a partial solution due to the motion of boundary nodes as follows:

$$[[K_{BB}] - [K_{BI}]\,[K_{II}]^{-1}[K_{IB}]]\,\{U_B\} = \{F_B\} - [K_{BI}]\,[K_{II}]\text{-}1\{F_I\}$$

i.e.,

$$[K_{BB}]^*\{U_B\} = \{F_B\}^* \qquad \textbf{(2-14)}$$

where,

$$[K_{BB}]^* = [K_{BB}] - [K_{BI}]\,[K_{II}]^{-1}[K_{IB}] \qquad \textbf{(2-15)}$$

$$\{F_B\}^* = \{F_B\} - [K_{BI}]\,[K_{II}]^{-1}\{F_I\}$$

The above equations can be repeatedly applied to different superelements. The complete system stiffness matrix is given by the following equation:

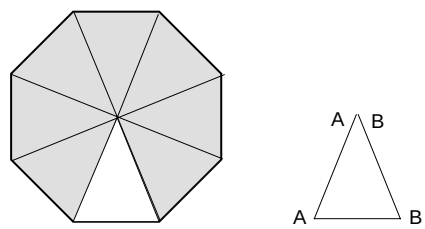
$$[K]_{system} = \sum_{i=1}^{nel} K_{el}^{i} + \sum_{j=1}^{nsel} \left[[K_{BB}^{j}] - [K_{BI}^{j}]\,[K_{II}^{j}]^{-1}[K_{IB}^{j}] \right] \qquad \textbf{(2-16)}$$

where *nel* is the number of elements in the main structure, and *nsel* is the number of superelements attached to the main structure. Once the displacements on the boundary or superelement nodes are computed, the displacements and hence stresses in the individual superelements can be recovered using equation (2-12).

Cyclic Symmetry

There are some structures such as impellers in a gas turbine, pressure vessels, and many others made up of identical segments yet do not have planes of reflective symmetry. Even though there is an axis of symmetry, they cannot be analyzed as axisymmetric structures since their geometry cannot be described as solids of

Figure 2-1. A Cyclic Symmetry Structure



revolution. This type of symmetry is known as *cyclic symmetry, sectorial symmetry, or rotational periodicity*. Instead of analyzing the entire structure, it is possible to model and analyze a representative substructure.

For a typical substructure shown in the figure above with interface boundaries A and B, the equilibrium equations can be written in the following form:

$$\begin{bmatrix} K_{II} & K_{IA} & K_{IB} \\ K_{IA}^T & K_{AA} & K_{AB} \\ K_{IB}^T & K_{AB}^T & K_{BB} \end{bmatrix} \begin{Bmatrix} D_I \\ D_A \\ D_B \end{Bmatrix} = \begin{Bmatrix} R_I \\ R_A \\ 0 \end{Bmatrix} + \begin{Bmatrix} 0 \\ F_A \\ F_B \end{Bmatrix} \quad \textbf{(2-17)}$$

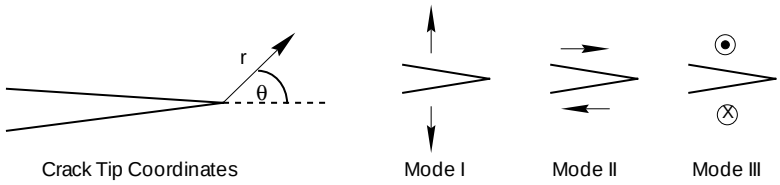
where $\{D_A\}$ and $\{D_B\}$ contain interface degrees of freedom along AA and BB respectively, and $\{D_I\}$ represents the interior degrees of freedom. The load vectors $\{R_A\}$ and $\{R_I\}$ represent applied loads on the substructure whereas the load vectors $\{F_A\}$ and $\{F_B\}$ result from elastic deformations and are applied along the AA and BB by the neighboring substructures. Since all repeating substructures are identical, $\{D_B\} = \{D_A\}$ and $\{F_B\} = -\{F_A\}$. Equation (2-17) reduces to:

$$\Gamma_i = \{\Phi\}_i^T [M] \{I_b\} \quad \textbf{(2-18)}$$

Linear Elastic Fracture Mechanics

The Basic System supports the analysis of linear elastic fracture mechanics problems for both 2D and 3D problems. There are three fundamental modes of crack opening as shown in the following figure. Mode-I represents the opening of the crack due to applied tension loading perpendicular to the plane of the crack whereas mode-II describes the solution around the crack due to shear loading parallel to the crack faces (in-plane shear). Mode-III describes the solution around the crack due to shear loading perpendicular the crack faces (anti-plane shear). Two different techniques are used to evaluate the Fracture Mechanics characteristics as follows.

Figure 2-2. Coordinates and Deformation Modes at the Crack Tip



Displacement Method

For mode-I, the stress and displacement fields in the neighborhood of a crack in crack tip coordinates (see figure above) have the following form (Irwin, 1961):

(2-19)

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{K_I}{(2\pi r)^{\frac{1}{2}}} \cos\frac{\theta}{2} \begin{Bmatrix} 1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2} \\ 1 + \sin\frac{\theta}{2}\sin\frac{3\theta}{2} \\ \sin\frac{\theta}{2}\cos\frac{3\theta}{2} \end{Bmatrix}$$

$$\begin{pmatrix} u \\ v \end{pmatrix} = \frac{K_I}{2G}\left(\frac{r}{2\pi}\right)^{\frac{1}{2}} \begin{Bmatrix} \cos\frac{\theta}{2}\left(3 - 4\nu - 1 + 2\sin\frac{\theta}{2}\right) \\ \sin\frac{\theta}{2}\left(3 - 4\nu + 1 - 2\cos\frac{\theta}{2}\right) \end{Bmatrix} \quad \textbf{(2-20)}$$

In the above equations, KI is known as the stress intensity factor for the opening mode. Each mode of fracture has a stress intensity factor dependent on the crack geometry and applied loading. The stress intensity factors are not related to stress concentration factors. The stress intensity factors not only describe the displacement and stress field in vicinity of a crack tip, but also determine the elastic energy release rates (G) associated with a crack. The relation between K factors and G is:

$$G = (1 - n^2)(K_{I2} + K_{II}^2)/E + K_{III}^2/E \quad \textbf{(2-21)}$$

J-Integral

For two dimensional homogenous models, along any curve S which is traversed in the counterclockwise direction between the two crack sides (Figure 2-3), the J-integral is defined as:

$$J = \int_s \left(w dx_2 - \rho_i \frac{\partial u_i}{\partial x_1} ds \right)$$

where:

$w = \int_o^\varepsilon \sigma_{ij} d\varepsilon_{ij}$ $\quad i, j = 1, 2, 3$ is the strain energy density, and

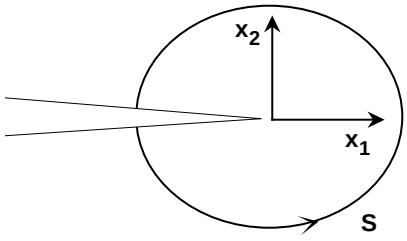
u_i = displacement in i direction

and,

ρ_i is the traction exerted on the body bounded by S and the crack surface

The J-integral vanishes in the absence of singularities. However, once there is a singularity such as a crack inside the path of integration, the integration becomes non-zero. The J-integral is a scalar measure of the state of straining at the crack tip. Characteristics of the crack such as the Stress Intensity Factor (K) are evaluated using the J-integral approach. It can be shown that this integration is path-independent and should be evaluated along a path chosen to give the greatest computational advantage.

Figure 2-3. A Crack Tip Enclosed by a J-integral Path



The above formulation is only applicable for plane strain and plane stress models with no internal body forces. For axisymmetric models and problems with thermal loads, additional terms should be added which include area integration of the fields inside the path.

Thermal Problems

In the presence of thermal loads, the J-integral is modified to the form:

$$J = \int_s \left(w dx_2 - \rho_i \frac{\partial u_i}{\partial x_1} ds \right) + \int_A \left(\beta \sigma_{ij} \delta_{ij} \frac{\partial T}{\partial x_1} dA \right)$$

where,

A	= Area inside the path
β	= Coefficient of thermal expansion
δ_{ij}	= Cronecker delta ($\delta_{ij} = 1$ if i=j, and $\delta_{ij} = 0$ if i≠j)
T	= Temperature

Axisymmetric Problems

For axisymmetric models, the J-integral is modified to the form:

$$J = \frac{1}{R_c} \left[\int_s R \left(w dx_2 - \rho_i \frac{\partial u_i}{\partial x_1} ds \right) - \int_A \left(w - \rho_\theta \frac{u_r}{R} \right) dA \right]$$

where,

R_c	= Crack tip radius
R	= Radius (distance of the point of integration from the axis of symmetry)
r, θ	= Radial and tangential directions
A	= Area inside the path

Main Features of J-Integration

1. Can be used for the rectangular (PLANE2D) and triangular (TRIANG) elements with or without midside nodes. Also available for plane strain, plane stress as well as axisymmetric models.
2. Can be easily defined for any circular path by specifying only a few parameters.

3. The path does not have to lie on the element edges and can pass through elements.
4. There is no limitation on the mesh formation (it can be generated as automatic mesh and does not have to conform to any symmetric patterns).
5. Evaluated entirely in the Stress module, therefore, can be modified and re-evaluated by rerunning of the stress module only.

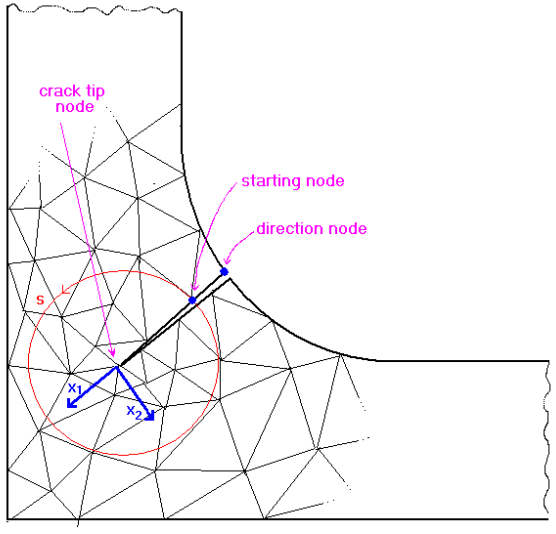
How to Define the Path

The paths are defined as circles centered at the crack tip. The following steps are required to define the path and other characteristics:

1. Activate the J-integral evaluation flag (in **A_STRESS**).
2. Define the crack tip node.
3. Define the starting node on the path. This node specifies the crack radius as well.
4. Define the direction of the crack by specifying a node on the crack face (could be the same as starting node).
5. Specify the type of crack (half or full circle).
6. Specify the number of integration points on the path.

The above path characteristics are specified by issuing a single command (**J_INTCRDEF**). The elements along the path as well as those inside (in the case of the area integration) are identified automatically.

Figure 2-4. Nodes which Define a J-integral Path



Notes:

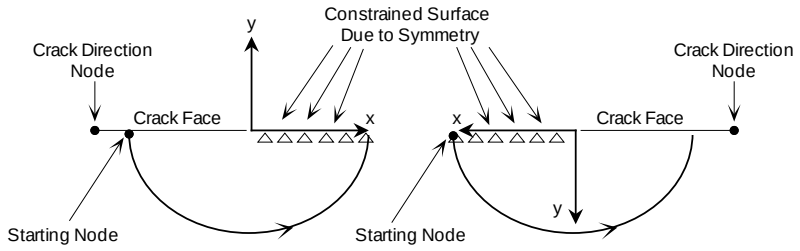
1. The path starting node should lie on the side of crack such that it would conform to the counterclockwise direction of the path.
2. The nodes on the two faces of the crack should not be merged in order to model the material rupture, due to the crack, properly.
3. You may define multiple cracks on the same model with more than one path for each crack. (Up to a total of 10 paths.)
4. Avoid passing the circular path through regions with high stress gradient or defining the path too close to the crack tip singularity.
5. In cases where the stress gradients are inevitably high, use more integration points. (You may wish to refine the mesh as well.)
6. You may use the J-integral in conjunction with the crack element (where it is available, i.e., higher order rectangular element). Using the crack element has the effect of increasing the accuracy of the calculated fields in the vicinity of the crack tip by moving the just-off-the-crack-tip-midside node to the element quarter points in order to produce the representative singularity conditions at the crack tip. (This increases the accuracy of the J-integration significantly where the path is close to the crack tip.)

Symmetric Modeling

If the problem is crack-axisymmetric, meaning that the model geometry, the boundary conditions as well as the applied loads are symmetric with respect to the crack axis, then only half of the model may be modeled with the appropriate symmetric boundary conditions. The J-integral in this case reduces to a half circle.

According to the convention described earlier and comparing the following two models, the path starting node does not always lie on the crack face when only half of the model is considered. However, the crack direction node always lies on the crack face.

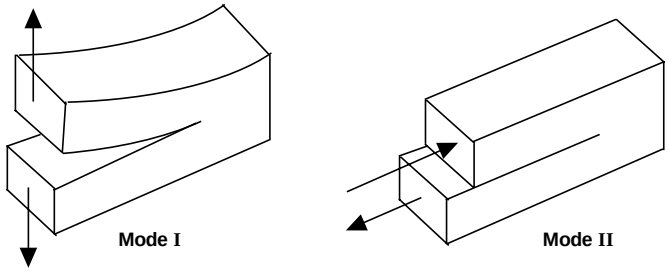
Figure 2-5. Two Different Models for a Half Circle J-integral Path



Evaluated Quantities

The combined J-integral value is evaluated as the sum of the individual values for each one of the two crack modes (for 2D problems): $J = J_I + J_{II}$

Figure 2-6. Modes of Cracking for Two Dimensional Problems



where J_I and J_{II} correspond to Modes I and II, respectively.

The corresponding Stress Intensity Factors can be evaluated according to:

$$K_I = \sqrt{\alpha E J_I}$$
$$K_{II} = \sqrt{\alpha E J_{II}}$$

where,

J_I, J_{II} = Modes I and II stress intensity factors

E = Modules of Elasticity

α { = 1 for plane stress models
= $1/1-\gamma^2$ for plane strain and axisymmetric models

γ = Poisson Ratio

In this case, due to symmetry, the J-integral is equal to J_I and J_{II} is equal to zero. Consequently, the output J is twice the value which is computed from the half circle path.

Notes:

1. Individual J-integral and K modes are always evaluated except for the thermal and axisymmetric problems.
2. For the crack-axisymmetric problems (whether the path is modeled as half or full circle), Mode II cracking is irrelevant. Therefore the combined J corresponds entirely to Mode I cracking.
3. From the above two statements, it is inferred that, for general thermal or axisymmetric problems, J and K cannot be evaluated for individual modes. However, for crack-axisymmetric models, the combined J corresponds to JI from which KI can be evaluated.
4. There are no limitations on the mesh type (whether it is crack-axisymmetric or not).

Modal Analysis

The computation of natural frequencies and mode shapes is known as modal or normal modes analysis. The finite element system of equations for dynamical systems can be written as:

$$[K]\{U\} + [C]\{\dot{U}\} + [M]\{\ddot{U}\} = \{F(t)\} \quad \textbf{(2-22)}$$

where [M] is the mass matrix, and [C] is the damping matrix. For free vibrations, the above equation takes the form:

$$[K]\{U\} + [C]\{\dot{U}\} + [M]\{\ddot{U}\} = 0 \quad \textbf{(2-23)}$$

When undamped linear elastic structures are initially displaced into a certain shape, they will oscillate indefinitely with the same mode shape but varying amplitudes. The oscillation shapes are called the mode shapes and the corresponding frequencies are called natural frequencies. The term modal analysis has been used throughout this manual for the study of natural frequencies and mode shapes. For undamped linear elastic structures, the above equation reduces to:

$$[K]\{U\} + [M]\{\ddot{U}\} = 0 \quad \textbf{(2-24)}$$

With no externally applied loads, the structure is assumed to vibrate freely in a harmonic form defined by:

$$U(t) = \phi \sin(\omega t + \theta) \quad \textbf{(2-25)}$$

which leads to the eigenvalue problem:

$$[[K] - \omega^2[M]]\ \{\phi\} = \{0\} \quad \textbf{(2-26)}$$

where ω is the natural frequency and ϕ is corresponding mode shape of the structure.

Eigenvalue Extraction Methods in Modal Analysis

There are four methods of computing the eigenvalues in the Basic System. They are:

- Subspace Iteration
- Lanczos Method
- Jacobi Method
- Inverse Power Method

These methods are fully discussed in the COSMOS/M Theoretical Manual.

Treatment of Rigid Body Modes in Modal Analysis - Shifting

Shifting can be viewed as a displacement of the origin in a plot of the eigenvalues. By appropriately selecting the shift points, the iteration process can be made to converge to any or all modes (including rigid body modes) of the structural system. The speed of convergence can be accelerated by shifting close to the root being sought, and it is therefore a good practice to shift at intervals during an iteration process. The shift option is applicable to both the subspace and inverse iteration methods.

Treatment of Rigid Body Modes in Modal Analysis - Soft Spring Addition

In addition to or in lieu of frequency shifting, COSMOS/M allows you to use the soft spring option for computing natural frequencies and mode shapes of structures with rigid body modes. As in static stress analysis, the soft spring stiffnesses are added to the diagonal terms of the stiffness matrix. However, the computed frequencies depend on the magnitude of the soft spring stiffness. In addition, the natural frequencies computed when using either the shift option or the soft spring addition, or a combination of both, may vary in each case, depending on the mag-

nitudes of shift or spring stiffness specified; see Chapter 9, *Performing Analysis*, for more information.

In-Plane Effects in Modal Analysis - Stress Stiffening and Softening

The effect of stress stiffening on structures in linear static analysis was discussed in an earlier section. The following is a continuation of this discussion for free vibration problems. In order to consider the in-plane effects on the natural frequencies of a structure, the structure *must* be first loaded and constrained for computing the geometric stiffness matrix. Therefore, a static stress analysis *precedes* the computation of natural frequencies with in-plane effects. In-plane loads have a stiffening or softening effect on the vibrational behavior of structures. Tensile membrane forces increase the natural frequencies where as compressive membrane forces decrease them. Similar to the static solution with in-plane effects, normal modes analysis in the presence of membrane forces is performed in two stages. In the first stage, the displacements $\{u_i\}$ for the *loaded* and *constrained* structure are computed using the conventional stiffness matrix [K]. In the second stage, the geometric stiffness matrix $[K_G(u_i)]$ is established based on the computed displacements $\{u_i\}$ and added to the conventional stiffness matrix [K], and then along with the mass matrix, the natural frequencies (ω) and the corresponding *virtual* displacements which are the eigenvectors, $\{\phi_{i+1}\}$, are solved. With the inclusion of in-plane effects, the finite element system of equations for computing the natural frequencies and mode shapes can therefore be written as:

$$([K] + [K_G(u_i)] - \omega^2 [M])\{\phi_{i+1}\} = \{0\} \quad \textbf{(2-27)}$$

where $[K_G(u_i)]$ is the geometric stiffness matrix computed based on $\{u_i\}$, [M] is the mass matrix, $\omega$is the natural frequency, and $\{\phi_{i+1}\}$ represents the eigenvectors which describe the natural modes of the structure.

The inclusion of in-plane effects in natural frequency computation is specified using the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command.

Eigenvalue Buckling Analysis

Buckling can be defined as the sudden deformation which occurs when the stored membrane energy is converted into bending energy with no change in the externally applied load. Buckling occurs when the total stiffness matrix becomes singular:

$$[K - \lambda K_G]\{q\} = \{0\} \quad \textbf{(2-28)}$$

where K_G is the geometric stiffness matrix (also referred to as *stress* stiffness matrix or *differential* stiffness matrix), and it is independent of the material properties of the structure in contrast to the conventional structural stiffness matrix. The multiplier λ represents the factor to the applied loads to cause buckling. The computation of buckling load factors is an eigenvalue problem, and all methods discussed earlier under modal analysis are applicable.

Modal Analysis of Piezoelectric Materials

The electromechanical constitutive equations for linear behavior are written as follows:

$$\{\sigma\} = [D]\{\varepsilon\} - [e]\{E_l\}$$
$$\{D_l\} = [e]^T\{\varepsilon\} + [\varepsilon_d]\{E_l\} \quad \textbf{(2-29)}$$

where $\{\sigma\}$ is the stress vector, $\{D_l\}$ is a vector of electric flux density, $[D]$ is the matrix of elastic coefficients at constant electrical field, $\{\varepsilon\}$ as before represents the mechanical strain, $[e]$ is a matrix of piezoelectric properties, $\{E_l\}$ is the vector representing the electric field, and $[\varepsilon^d]$ is the dielectric property matrix at constant mechanical strain. The elastic, dielectric, and piezoelectric material properties and the procedure for their input are fully explained in Chapter 6, *Material, Sectional, and other Physical Properties*. Without the coupling term provided by the piezoelectric matrix $[e]$, the above equations represent the constitutive equations for elastostatic and electrostatic problems.

The finite element system of equations for the computation of natural frequencies and mode shapes have the same form as that of the modal analysis. The eigenvalue extraction methods in piezoelectric modal analysis are the same as those used in modal analysis.

Guyan Reduction

For very large problems, computing eigenvalues and eigenvectors requires a lot of time and computational effort. However, condensation in the form of Guyan Reduction can be used to reduce the number of degrees of freedom significantly, thus, increasing the speed of the program substantially. In COSMOS/M the Guyan Reduction can be used with the Subspace and Jacobi eigenvalue extraction schemes whereby the number of the equations is reduced by eliminating degrees of freedom of lesser importance. Let D_m represent the more important or master degrees of freedom, and D_s represent degrees of freedom of lesser importance or slave. The equations of motion can then be written as:

$$\left(\begin{bmatrix} K_{mm} & K_{ms} \\ K_{ms}^T & K_{ss} \end{bmatrix} - \omega^2 \begin{bmatrix} M_{mm} & M_{ms} \\ M_{ms}^T & M_{ss} \end{bmatrix} \right) \begin{Bmatrix} D_m \\ D_s \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \quad \textbf{(2-30)}$$

The basic assumption in the Guyan Reduction is that for lower frequency modes, inertia forces due to slave d.o.f. are much less important than elastic forces transmitted by the master d.o.f. This criteria must be used in selecting which d.o.f. should be master and which ones should be slave. With this assumption in mind, the lower part of the mass matrix can be ignored and we find from the lower portion of the matrix:

$$\{D_s\} = -K_{ss}^{-1} K_{ms}^T D_m \quad \textbf{(2-31)}$$

The slave degrees of freedom are then eliminated from the upper set of equations to yield:

$$([K_{eq}] - \omega_2 [M_{eq}])\{D_m\} = \{0\} \quad \textbf{(2-32)}$$

where,

$$K_{eq} = K_{mm} - K_{ms} K_{ss}^{-1} K_{ms}^T \quad \textbf{(2-33)}$$

$$M_{eq} = M_{mm} - M_{ms} K_{ss}\text{-}1 K_{ms}^T \qquad \text{(or lump mass at master d.o.f.)}$$

From the above equation, it can be concluded that K_{eq} and M_{eq} are in general full matrices, therefore, *m* (no. of master d.o.f.) must be considerably smaller than the total d.o.f. for Guyan reduction to be cost effective. If the matrix structure of the problem has small bandwidths, it may be more advantageous to avoid condensation.

How to Use Guyan Reduction in COSMOS/M

1. Model your problem as usual.
2. Select master d.o.f. by using commands in the "MASTER DOF" submenu in LoadsBC > STRUCTURAL.
3. Create MASS elements at locations of master degrees of freedom using the **M_PT** (Meshing > PARAMETRIC MESH > **Points**) or **EL** (Meshing > ELEMENTS > **Define Element**) commands. If you use the **EL** command, remember to use keypoints as the associated geometric entity. Estimate the associated masses and enter the information using the **RCONST** (Propsets > **Real Constant**) command.
4. Use the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command to specify Guyan reduction with Jacobi (GJ) or Subspace (GS).
5. Run frequency analysis using the **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) command. The frequencies and mode shapes will be available in the database.

Verification problem F27A.GEO illustrates the proper sequence of commands pertinent to the Guyan Reduction technique.

ASME Section III Stress Evaluation

An evaluation of stress results, in a form suitable for comparison with the ASME code section III requirements, is formulated. Stress resultants, membrane, bending, peak, and stress intensities are computed based on an arbitrarily defined cross-sectional.

The failure criteria for typical FEA include the peak stresses. Allowable stresses Sm, 1.5 Sm, and 3.0 Sm, specified by the ASME code section III are based on redundant forces and moments along a section associated with shell-type structure (Gordon, 1976). In the formulation presented here (Kroenke, 1974), the

stresses are linearized along a "stress classification line" (or so called "section") in an axisymmetric model and the primary plus secondary and peak stresses are evaluated (at the two ends of the section) in a form which is convenient for

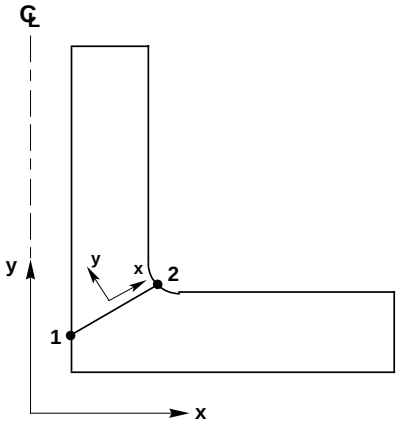



comparison with the allowable stresses (i.e., Sm, 1.5 Sm and 3 Sm) specified by the ASME code and in accordance with Gordon, 1976 and Kroenke, 1974.

The in-plane bending moment, membrane and shear forces, as well as the out-of-plane membrane forces and bending moments are evaluated first along a section, from which the peak stresses are calculated for two types of models: plane stress/strain and axisymmetric problems.

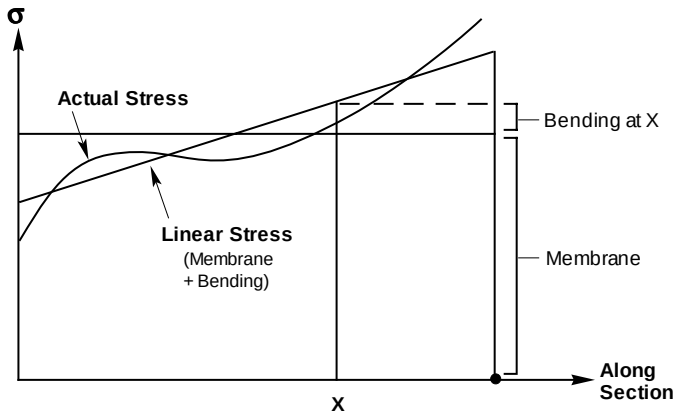
A section is defined by a line passing through the thickness and is specified by nodes at its two ends.

Figure 2-7. Section Defined Through the Thickness and the Corresponding Section Coordinates



The section must present a rational plane of bending which, in most cases will be perpendicular to both surfaces and the mid-plane. For most structures with parallel surfaces, such as pipes, shells and external nozzles, this criteria can be easily met. However, in irregular areas such as in the nozzle to the shell juncture, rational planes of bending may be specified to be perpendicular to the mid plane and have the same angle as between the section and the surface on both sides. The linearized stresses at the two ends are calculated as the sum of the membrane (primary) stresses which are constant along the section and the bending stresses which vary linearly along the section. These stresses are calculated from the internal forces and moments on the section. The evaluation procedure of these stresses is described in Appendix C.

Figure 2-8. FEA Calculated Stresses (Actual Stress) and the Linearized Stresses Along a Section



The "peak" stress value at a point along the section is defined as the difference between the actual stress calculated by the FEA and the linearized (membrane plus bending) stress. Therefore the "peak stress" at the two ends and the center of the section are:

$$\sigma p_{i1} = s_{i1} - \sigma^{m}_{i} - \sigma^{b}_{i1} \quad \text{(at end 1)}$$

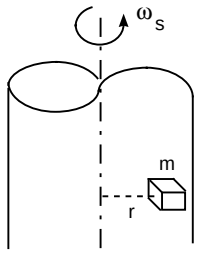
$$\sigma p_{i2} = s_{i2} - \sigma^{m}_{i} - \sigma^{b}_{i2} \quad \text{(at end 2)} \qquad \textbf{(2-34)}$$

$$\sigma p_{ic} = s_{ic} - \sigma^{m}_{i} \quad \text{(at the section center)}$$

Where i corresponds to the i-th component of stresses. σ^{m}_{i} corresponds to the constant membrane along the section and σ^{b}_{i1} and σ^{b}_{i2} correspond to bending stresses at the two ends. Note that the bending stress at the center of the section is considered to be zero.

Spin Softening

The Small Deflection theory of Linear Static analysis can not directly account for large changes in geometry due to the centrifugal loads. Therefore the effect can be accounted for by an adjustment of the stiffness matrix, called spin softening. To understand the application of this effect, consider the following spinning shaft.

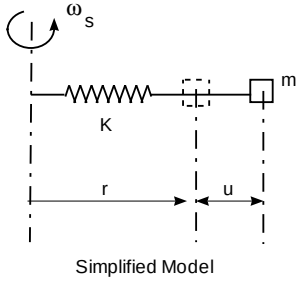


For an infinitesimal element mass (m) inside the shaft the equilibrium of force without accounting for the effect of the radial deflection (small deformation theory) requires:

$$Ku = mr\omega_s^2 \quad \textbf{(2-35)}$$

where:

$mr\omega_s^2$= Centrifugal force on the mass
K = Stiffness
r = Radial distance of the mass
u = Radial deflection of the mass
ω_s = Angular spin velocity



Simplified Model

Now if the deflection is large enough to not be ignored in evaluation of centrifugal forces, the equilibrium force requires:

$$Ku = m\,(r+u)\,\omega_s^2 \quad \textbf{(2-36)}$$

where the centrifugal force $[m\,(r+u)\,\omega_s^2]$ is evaluated at its final resting position. Rearranging terms, lead to:

$$(K - m\omega_s^2)u = mr\omega_s^2 \quad \textbf{(2-37)}$$

Defining:

$$K_s = K - m\omega_s^2 \quad \textbf{(2-38)}$$

From equations (2-37):

$$K_s\, u = mr\omega_s^2 = F_s \quad \textbf{(2-39)}$$

Comparing equation (2-39) with equation (2-35), it is clear that K should be reduced in magnitude by the amount of $m\omega_s^2$ (i.e., spin softening) to account for the large deformation effect in the plane of rotation.

The above formulation can be extended to frequency analysis, where the eigenvalue problem is represented by:

$$[K_s - \omega^2 M]\{\phi\} = 0 \qquad \textbf{(2-40)}$$

where ω is the natural frequency of the spinning body.

When the spin softening is considered in the solution, it is often useful to activate the stress stiffening effect as well (see note below).

In COSMOS/M only the spin softening effect is considered in frequency analysis of rotating bodies. It is not incorporated into Static analysis, since the NSTAR program can be used more properly to solve problems with large deformation.

For a general 3D problem with angular velocity in different direction, the reduced stiffness can be represented by:

$$K_{xx}^s = K_{xx} - M_{xx}(\omega_y^2 + \omega_z^2)$$

$$K_{yy}^s = K_{yy} - M_{yy}(\omega_x^2 + \omega_z^2) \qquad \textbf{(2-41)}$$

$$K_{zz}^s = K_{zz} - M_{zz}(\omega_x^2 + \omega_y^2)$$

$$K_{xy}^s = K_{xy}, \quad K_{yz}^s = K_{yz}, \quad K_{xz}^s = K_{xz},$$

where, K_{xx}, K_{yy}, and K_{zz} are the element stiffness calculated in x, y, and z directions, K_{xx}^s, K_{yy}^s and K_{zz}^s, are the reduced stiffnesses used in place of the element stiffnesses, M_{xx}, M_{yy}, and M_{zz}, are the element masses and ω_x,ω_y, and ω_z are the angular velocities about the x, y and z axis.

To activate stress stiffening effect, turn on the centrifugal loading option flag in **A_STATIC** command together with In-plane Load flag in **A_FREQ** command.

Inertia Relief

In Linear Static analysis, the external forces on an elastic body can be balanced by inertia forces induced by a translational acceleration field.

(2-42)

$$\{F\}_t + \int_v \{a\}_t \rho \, dv = 0$$

where:

$\{F\}_t$ = Applied external forces

$\{a\}_t$ = Translational acceleration due to inertia (to be calculated)

ρ = Material density

v = Material volume

The above equation may be represented as:

$$\begin{Bmatrix} F_x \\ F_y \\ F_z \end{Bmatrix}_t + \begin{bmatrix} M_{xx} & 0 & 0 \\ 0 & M_{yy} & 0 \\ 0 & 0 & M_{zz} \end{bmatrix} \begin{Bmatrix} a_x \\ a_y \\ a_z \end{Bmatrix}_t = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix} \quad \textbf{(2-43)}$$

where M_{xx}, M_{yy} and M_{zz} are total masses in x, y and z directions and are evaluated by a row-by-row summation of the elemental mass matrices over translational degrees of freedom. Once M_{xx}, M_{yy} and M_{zz} are evaluated, equation (2-43) yields the induced accelerations a_x, a_y and a_z which then will be applied as an acceleration field to the FEA solution.

✍ Only translational field is considered in the above field and any induced rotation is ignored.

Mode Correlation and Checking

The MODE_CHECK (Analysis, Frequency/Buckling, Mode_Check Option) command provides several options that can be very useful in checking modes obtained from experimental (secondary) data against primary modes resulting from FEA. The secondary data can also come from a different FEA run.

The command provides 3 options:

6. Calculate cross-correlation,
7. Check mode orthogonality, and
8. Evaluate modal assurance neglecting the effect of mass.

Cross-Correlation

While frequencies obtained from experimental data are recognized as the most reliable, frequencies from FEA are considered accurate if they are within 5% discrepency. In general, the frequencies are far less sensitive to structural changes than mode shapes. The quality of the secondary modes is evaluated based on their observance of the orthogonality with respect to the primary mass matrix as illustrated below:

Let $[\Phi_P]$ represent the matrix of the primary mode shapes where each column represents a mode shape. Similarly, let $[\Phi_S]$ represent the matrix of the secondary mode shapes. The mass matrix of the primary system will be denoted by $[M_P]$.

The product $[O_{PP}] = [\Phi_P]^T[M_P][\Phi_P]$ will always generate a diagonal matrix. Now if the secondary data are exact, the product $[O_{PS}]=[\Phi_P]^T[M_P][\Phi_S]$ will generate the same diagonal matrix. In practice $[O_{PS}]$ is not expected to be equal to $[O_{PP}]$ or even to be a perfect diagonal matrix. Instead, $[O_{PS}]$ will have off-diagonal terms. However, the degree of *how diagonal* this product is, represents a tool for identifying and assessing the quality of correspondence between the primary and secondary modes. Off-diagonal terms smaller than 10% of the diagonal are generally considered acceptable in practice.

For mass-normalized mode shapes, the dominant terms of $[O_{PS}]$ should have magnitudes close to unity to indicate similarity. The sign is insignificant when comparing mode shapes. +1 or -1 indicate perfect similarity.

The statement in the above paragraph assumes that $[\Phi_S]$ contains an ordered set of modes with one-to-one correspondence with $[\Phi_P]$. Obviously this may or may not be the case since $[\Phi_S]$ may contain a different number of modes than $[\Phi_p]$ and the order of the modes may be different. In other words, Φ_{S1}, the first column (mode) in $[\Phi_S]$ may or may not belong to Φ_{p1}. It follows that the product $[O_{PS}]=[\Phi_P]^T[M_P][\Phi_S]$ may or may not be diagonal or square, even when some modes match perfectly. However a dominant term O_{Ps} ( ) indicates matching between primary mode l and secondary mode k. A dominant term is identified if other terms are less than 10%. The presence of more than one dominant term may indicate multiple matching. For example if the columns of $[\Phi_S]$ are identical and they match primary mode l, all the terms of row l of O_{PS} will be equal, but other rows should have no dominant terms. It should also be noted that the dominant terms can be negative since multiplying a mode by (-1) does not change its signature.

- Before carrying matrix multiplications, the program modifies the secondary matrices to correspond to the same degrees of freedom used by the primary system.

- If the normalization flag in the command is activated, the dominant terms should be around unity for a mode matching.

Orthogonality of Secondary Modes

To evaluate the integrity of the secondary modes, it is important to check their orthogonality with respect to the primary mass matrix. In other words, the product $[\Phi_s]^T[M_P][\Phi_s]$ should be dominantly diagonal (off-diagonal terms within 10% of the diagonal). If the secondary modes do not pass this test, a thorough re-examination of the test data and the FEA model should be carried out. For mass-normalized mode shapes, the magnitudes of dominant terms of $[O_{PS}]$ should be close to unity to indicate similarity.

Cross Modal Assurance Criterion

The modal assurance criterion is used to check the independence of two modes. Basically, it is similar to cross-correlation but it ignores the effect of the system mass. The cross modal assurance criterion matrix [M] is computed from:

$$[M] = [[\Phi_P]^T[\Phi_p]x[\Phi_s]^T[\Phi_s]]/[\{D_{pp}\}\{D_{ss}\}^T]$$

Where:

x denotes element-by-element multiplication,

/ denotes element-by-element division,

$\{D_{pp}\}$ represents the vector of diagonal terms of $[\Phi_p]^T[\Phi_p]$, and

$\{D_{ss}\}$ represents the vector of diagonal terms of $[\Phi_s]^T[\Phi_s]$

Interpolation

In performing any of the mode-check options described above, the secondary data need to be modified to correspond to the primary model. There are two options available in the Mode_Check command for preparing the secondary data. The first option is to use linear interpolation of the secondary data at the primary grid locations. Extrapolation may also be used in some cases. The second option uses the primary grid point of the closest mass.

The normalization flag should always be activated unless the secondary data is obtained from an FEA program using the same grid used for the primary model.

To check/identify mode shapes:

1. Run frequency analysis on the current model to calculate the desired number of modes. The primary mode shapes are now defined.
2. If you are using experimental data or modes resulting from an FEA program other than GEOSTAR for the secondary data, write the information in the specified format. Refer to the online help for the Mode_Check command for details.
3. If you are using another GEOSTAR database, run frequency analysis. Close the secondary database and re-open the primary database in GEOSTAR. Then select **Analysis > FREQUENCY/BUCKLING >Mode check option** and select the desired task specifying the secondary database.
4. Run frequency analysis on the primary model. The results of the mode check will be written in the output file.

- It is recommended to run the modal assurance criterion first to evaluate the orthogonality of the secondary modes with respect to the primary mass. If the secondary data passes this test successfully, the other two options may be tried. Otherwise, a re-evaluation of the test data as well as the primary model is needed before proceeding.

- When you run frequency analysis while the check option is set to an option other the (0: Off) option, the program performs only the check, it does not calculate the frequencies. To return to the regular mode, issue the MODE_CHECK command again and select the default option.




3 Getting Started

Introduction

The previous chapters presented some of the finite element analysis and modeling features in the Basic System and their mathematical background. In this chapter, you will be presented with a hands-on example which walks you through a linear static analysis. A simple well designed drawing or illustration can indeed explain a concept that may otherwise take several paragraphs to describe, and this chapter will attempt to substantiate that. Moreover, you will see how simple it is to create and analyze models in COSMOS/M. A simple model geometry has been selected for linear static analysis using a compatible mesh as well as a non-compatible mesh to illustrate the bond feature.

Linear Static Stress Analysis

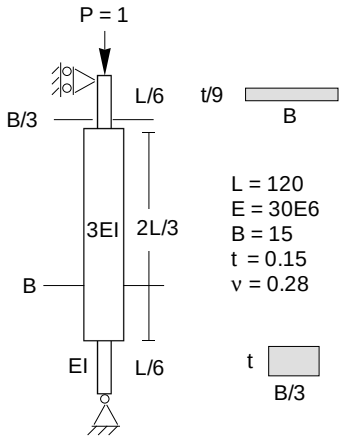
The figure below shows the geometry of the model for analysis with the applied loads and boundary conditions. The problem represents a beam with a stepped cross section, and it will be modeled with plane continuum elements (PLANE2D) only to show some features of the STAR module. Otherwise, this problem can be accurately modeled using beam elements in the Basic System.

To start with, enter GEOSTAR and provide a job name for this example. In order to proceed with geometry creation, execute the following steps to change the view and to define a working plane. Using the View Direction option icon (Binocular), select the second icon for X-Y view. Next, from Geometry > GRID, select the **Plane** command and provide the following input in the dialog box (defaults in this case):

Geo Panel: Geometry > GRID > **Plane**
Rotation/Sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line style > **Solid**

Geo Panel: Geometry > GRID > **Grid On**
Accept defaults...

Figure 3-1. Problem for Analysis



Problem Geometry and Cross Section Data

From Geometry > SURFACES, select **Draw w/ 4 Coord** (for creating a surface by either snapping four coordinates on the grid or by typing them in the dialog box). *Accept* the following input (you may use the mouse directly to create the surface or use the dialog box):

Geo Panel: Geometry > SURFACES > **Draw w/ 4 Coord**
Surface > **1**
XYZ-Coordinate of Keypoint 1 > **55,0,0**
XYZ-Coordinate of Keypoint 2 > **60,0,0**
XYZ-Coordinate of Keypoint 3 > **60,20,0**
XYZ-Coordinate of Keypoint 4 > **55,20,0**

Similarly, create surfaces 2 and 3 using the following coordinates in the dialog box:

Surface > **2**
XYZ-Coordinate of Keypoint 1 > **50,20,0**
XYZ-Coordinate of Keypoint 2 > **65,20,0**
XYZ-Coordinate of Keypoint 3 > **65,100,0**
XYZ-Coordinate of Keypoint 4 > **50,100,0**

Surface > **3**
XYZ-Coordinate of Keypoint 1 > **55,100,0**
XYZ-Coordinate of Keypoint 2 > **60,100,0**
XYZ-Coordinate of Keypoint 3 > **60,120,0**
XYZ-Coordinate of Keypoint 4 > **55,120,0**

The top and bottom sections which have the same dimensions will be modeled with the same number of elements.

Use the Auto Scaling icon (**SCALE;**) to properly view the created surfaces. For this problem, we will use the PLANE2D element to capture the in-plane behavior due to the applied load. Since the thickness of the top and bottom surfaces is different from that of the mid-section, we need two section (real) constant groups. When the finite element mesh is generated, the active element group number and the active real constant set are associated with the generated elements. Execute the following commands to define the element group, the material properties, and the first real constant set. From Propsets, select **Element Group** (**EGROUP**) command and specify PLANE2D (area type) element under group number 1. *Accept* all defaults in the next dialog box. From the Propsets menu, select **Pick Material Lib** (**PICK_MAT** command) and *Accept* all defaults. Alloy Steel (A_STEEL) in FPS units will be selected for material group number 1. Next, from the same menu Propsets, select **Real Constant** (**RCONST** command), accept default options for the first two dialog boxes and in the last, provide the thickness input as follows:

Geo Panel: Propsets > **Real Constant**
Associated element group > **1**
Real constant set > **1**

Start loc. of real const. > **1**
No. of real const. to be entered > **2**

RC1: Thickness > **0.15**
RC2: Material Angle (Beta) > **0.0**

You can now proceed to generate the finite element mesh for the top and bottom surfaces as follows. From Meshing > PARAMETRIC MESH, select **Surfaces** (**M_SF** command) and furnish the following input in the dialog boxes:

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **1**
Ending surface > **3**
Increment > **2**
Number of nodes per element > **4**

Number of elements on 1st curve > **4**

Number of elements on 2nd curve > **8**

Spacing ratio for 1st curve > **1.0**
Spacing ratio for 2nd curve > **1.0**

Before meshing the mid surface, define a second real constant set using **Real Constant** from Propsets, and provide the following input in the dialog boxes:

Geo Panel: Propsets > **Real Constant**
Associated Element Group > **1**
Real Constant set > **2**

Start loc. of real const. >**1**
No. of real const. to be entered > **2**

RC1: Thickness > **0.016667**
RC2: Material Angle (Beta) > **0.0**

Next, generate the finite element mesh of the mid-section as follows. From Meshing > PARAMETRIC MESH, select **Surfaces** (**M_SF** command) and type the following input in the dialog boxes:

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **2**
Ending surface > **2**
Increment > **1**
Number of nodes per element >

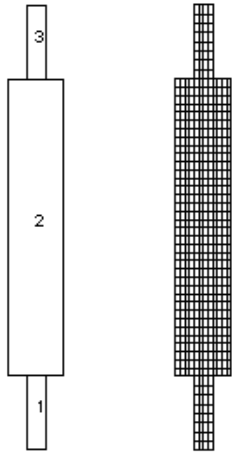
Number of elements on 1st curve > **12**

Number of elements on 2nd curve > **36**

Spacing ratio for 1st curve > **1.0**
Spacing ratio for 2nd curve > **1.0**

The figure shows the surface plots as well as the generated finite element mesh.

Figure 3-2. Surface and Finite Element Mesh Plots



The displacement constraints are enforced as follows: at the top support, the x-component of displacement is restrained whereas at the bottom support, both x- and y-components are fixed. These boundary conditions at the nodes are enforced as follows. From LoadsBC > STRUCTURAL > DISPLACEMENT, select **Define Nodes** (**DND** command) and provide the following input in the dialog box:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
Beginning node > **88**
Displacement Label > **UX: X translation**
Value > **0.0**
Ending node > **88**
Increment > **1**

Accept the defaults for the next dialog box. Note that the flow between the dialog boxes is recursive which means that you can also input the boundary conditions for node 3 as follows:

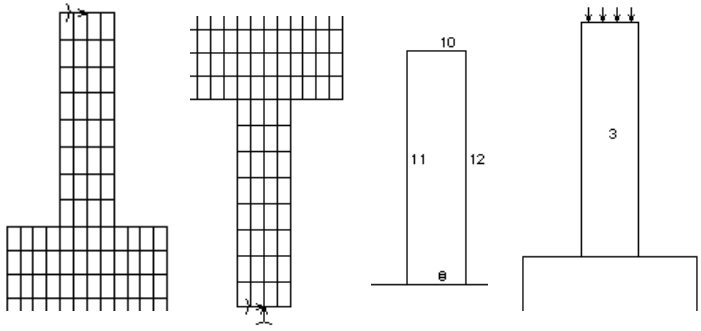
Beginning node > **3**
Displacement Label > **UX: X translation**
Value > **0.0**
Ending node > **3**
Increment > **1**

Displacement Label > **UY: Y translation**

After you *Accept* the last input (UY), you will again see the first dialog box for displacement boundary conditions. Click on *Cancel* to terminate the command.

The applied displacement boundary conditions are shown in the figure below.

Figure 3-3. Applied Boundary Conditions and Loads



The vertical load is applied as pressure loading on curve 10 shown in the figure above. From LoadsBC > STRUCTURAL > PRESSURE, select **Define by Curves** (**PCR** command) and provide input in the dialog box as follows:

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define by Curves**
Beginning curve > **10**
Pressure Magnitude > **1**
Ending curve > **10**
Increment > **1**
Pressure at the end of direction > **1**
Pressure Direction > **Normal Direction**

Similar to the previous operation, this command is also recursive. Therefore, after you *Accept* the above input, click on *Cancel* to terminate the command.

Since the mesh for each surface is generated independent of the neighboring one, there will be duplicate definition of nodes on the common boundary. These nodes need to merged in order to satisfy compatibility as follows. From Meshing > NODES, select **Merge** (**NMERGE** command) and *Accept* all defaults in the dialog box. The numbering gaps in the nodes as a result of merging is removed by using Edit > COMPRESS > **Nodes** (**NCOMPRESS** command). There will be 10 nodes compressed for this problem.

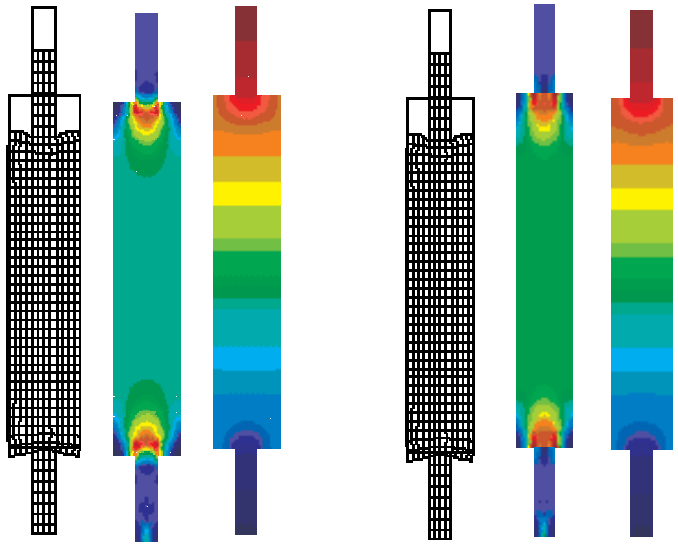
The finite element model is now ready for analysis. Use the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) to perform a linear static stress analysis. The results of analysis are written to the output file with extension OUT.

The deformed shape plot of the model can be viewed using the command **DEFPLOT** (Results > PLOT > **Deformed Shape**). You can superimpose the curve plot to better visualize the deformed state using the **CRPLOT** (Edit > PLOT > **Curves**) command.

To visualize the stress and displacement contours, use the following commands respectively: from Results > PLOT, select **Stress** (**ACTSTR** and **STRPLOT**) and from Results > PLOT, select **Displacement** (**ACTDIS** and **DISPLOT**). When you execute these commands with default options, you will be processing von Mises stresses and resultant displacement contours.

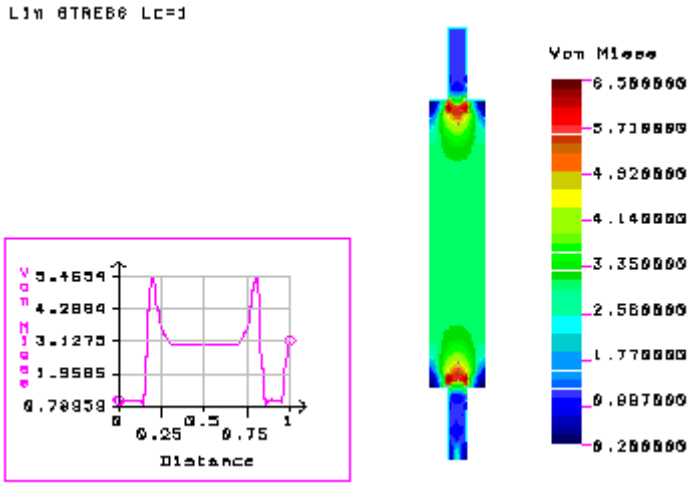
The figures below show the resultant displacement and von Mises stress contours as well the deformed shape plots of the model. The plots on the left half refer to the analysis results you just performed. On the right are the results from an analysis using the bond feature (see Chapter 8 and 11) in which the nodes at the common boundary curves were deliberately mismatched for verification purposes. As you can notice, the results from the two analysis appear to be identical.

Figure 3-4. Deformed Shape and Contour Plots of Compatible and Non-Compatible Models



You can try other postprocessing features for this problem. We will plot a line section along the center line from top to bottom of the von Mises stress contours. To start with, we will deactivate the display of the chart using the following procedure. From Results > PLOT, select the **Path Graph** (**LSECPLOT**) command and pick nodes from top to bottom along the center line of the model. When you reach the last node, click twice on that node to terminate the command. Note that you may need to use the Translate icon to position the contour plot away from the X-Y graph. As shown in the following figure, you will see the variation of the plotted result component along the line you just traced.

Figure 3-5. Line Section Graph for von Mises Stress Plot



4 Input Data Requirements

Introduction

Proper modeling and analysis specifications are crucial to the success of any finite element analysis. Irrespective of the type of analysis (structural, thermal, fluid flow, etc.), numerical solution using finite element analysis requires complete information of the model under consideration. The finite element model you submit for analysis must contain all the necessary data for each step of numerical simulation - geometry, elements, loads, boundary conditions, solution of system of equations, visualization and output of results, etc. This chapter attempts to conceptually illustrate the procedure for building a complete model for the three types of analysis you can perform in the Basic FEA System - linear static, buckling, and modal analysis.

Since a major portion of the effort in building a finite element model is in geometry creation and meshing, these topics will not be discussed here. The COSMOS/M User Guide (Volume 1) presents in-depth information on the procedures for model building and postprocessing in GEOSTAR. This chapter therefore only outlines those commands which are relevant for *analysis* in the Basic System. Chapter 10 presents some information on postprocessing features applicable to analysis in the Basic System.

For a detailed description of all commands, please refer to the COSMOS/M Commands Manual (Volume 2).



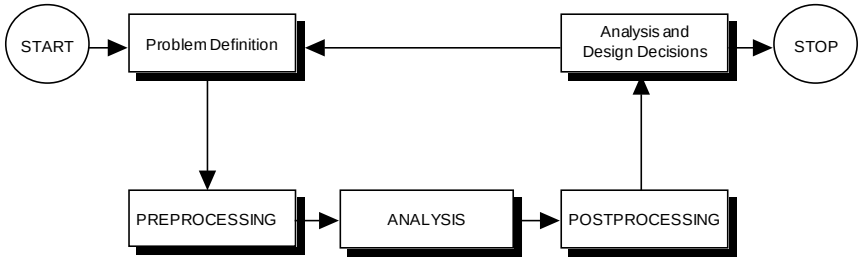

Modeling and Analysis Cycle in the Basic System

The basic steps involved in a finite element analysis are:

- Create the problem geometry.
- Mesh the defined geometry with appropriate type of element(s).
- Apply constraints on the finite element model.
- Define the loads on the model.
- Define the material and sectional properties.
- Submit the completed finite element model for analysis.
- Interpret and analyze the results.

These steps can be schematically represented as shown in the figure below.

Figure 4-1. Finite Element Modeling and Analysis Steps



Preprocessing refers to the operations you perform such as defining the model geometry, mesh generation, applying loads and boundary conditions, and other operations that are required *prior* to submitting the model for analysis. The term *analysis* in the above figure refers to the phase of specifying the analysis options and executing the actual analysis. Postprocessing refers to the manipulation of the analysis results for easy understanding and interpretation in a graphical environment.

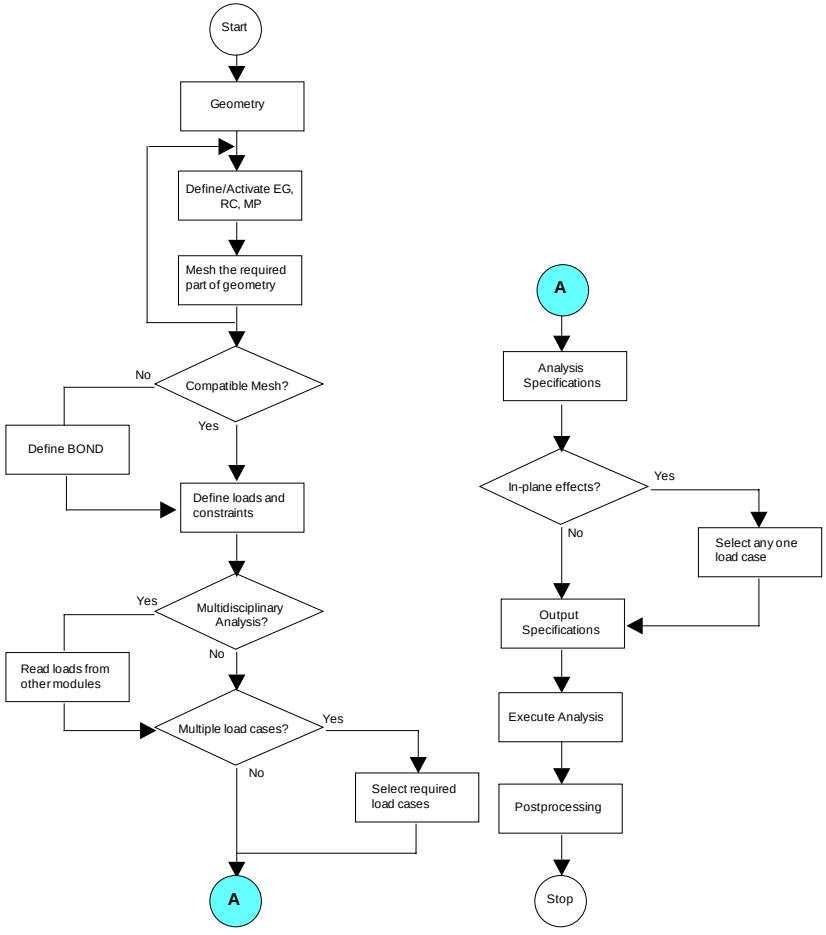
As outlined in Chapter 1, *About the Basic System…*, each analysis type in the Basic System has many features. Building the finite element model for a particular problem therefore depends on how you can utilize these features in an optimal and efficient way to yield the most accurate solution for the finite element model.

Flow Charts and Commands for Analysis in the Basic System

In the following pages, you are presented with flow charts of each analysis type and the corresponding commands or menu in a tabular form to assist you in developing your finite element model efficiently.

Linear Static Analysis

Figure 4-2. Flow Chart for Linear Static Analysis in the Basic System



Important Commands for Linear Static Analysis

The commands tabulated below provide you with information on the input of element groups, material and sectional properties, loads and boundary conditions, analysis specifications, and output specifications.

Table 4-1. Important Commands for Linear Static Analysis

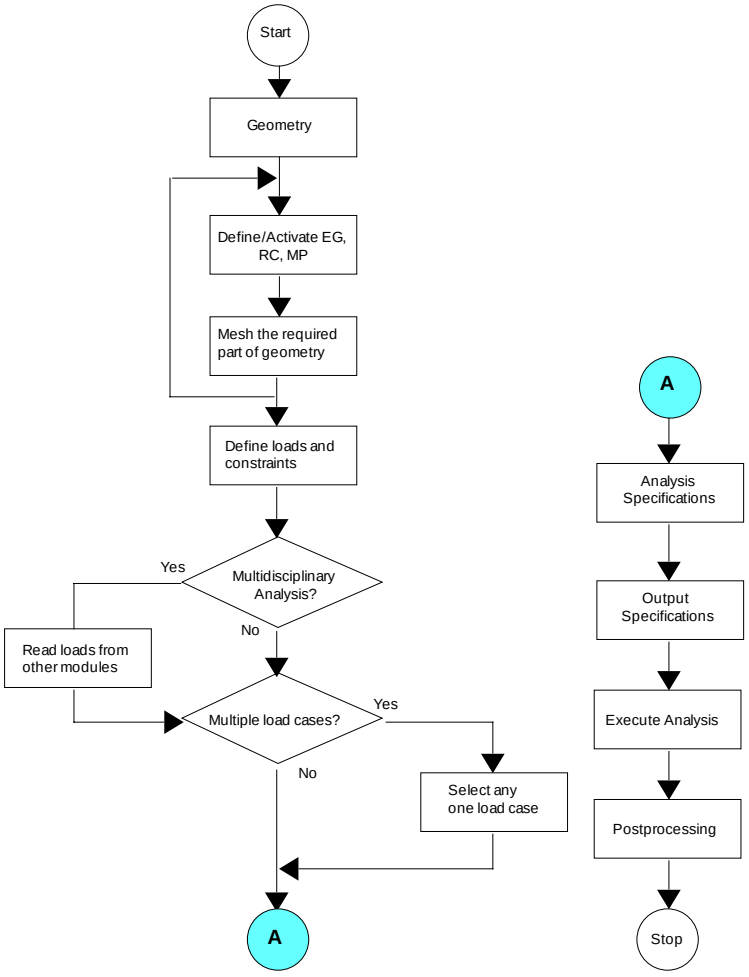
Function	COSMOS/M Menu	Command
Property definition	Propsets	EGROUP, MPROP, RCONST, PICK_MAT, USER_MAT, PICK_SEC, BMSECDEF, EPROPSET, EPROPCHANGE
Loads	LoadsBC > STRUCTURAL > DISPLACEMENT > FORCE > PRESSURE > GRAVITY > THERMAL > TEMPERATURE	. . . . . . D_ commands for displacements F_ commands for forces P_ commands for pressure Gravity load commands . . . NT_ commands for temperatures
Boundary conditions	LoadsBC > STRUCTURAL > DISPLACEMENT > COUPLING > BONDING	. . . . . . D_ commands for displacements CP_ commands for coupling BOND_ commands for bonding
Model verification	Meshing > ELEMENTS Analysis	. . . ECHECK DATA_CHECK, R_CHECK
Specifying analysis options	Analysis > STATIC	. . . LCSET, ADAPTIVE, P_ORDERLABS, A_STATIC, A_STRESS, STRESS
Specifying output options	Analysis > OUTPUT OPTIONS	. . . PRINT_OPS
Executing linear static analysis	Analysis > STATIC	. . . R_STATIC, R_STRESS
Postprocessing	Results	Refer to COSMOS/M User Guide (V.1) for more details.

* Please refer to Appendix B, *Command Summary*, for locating the path to menus from cryptic commands.




Buckling Analysis

Figure 4-3. Flow Chart for Buckling Analysis in the Basic System



Important Commands for Buckling Analysis

The commands tabulated below provide you with information on the input of element groups, material and sectional properties, loads and boundary conditions, analysis specifications, and output specifications.

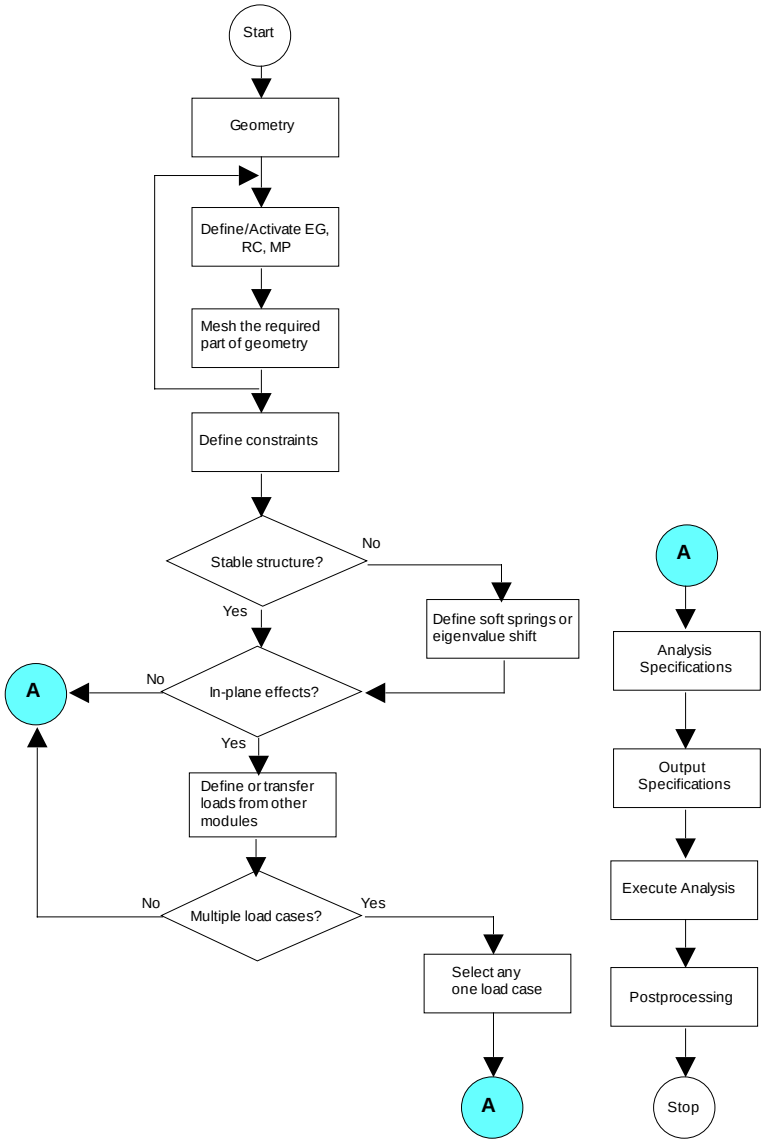
Table 4-2. Important Commands for Buckling Analysis

Function	COSMOS/M Menu	Command
Property definition	Propsets	EGROUP, MPROP, RCONST, PICK_MAT, USER_MAT, PICK_SEC, BMSECDEF, EPROPSET, EPROPCHANGE
Loads	LoadsBC > STRUCTURAL > DISPLACEMENT > FORCE > PRESSURE > GRAVITY > THERMAL > TEMPERATURE	. . . . . . D_ commands for displacements F_ commands for forces P_ commands for pressure Gravity load commands . . . NT_ commands for temperatures
Boundary conditions	LoadsBC > STRUCTURAL > DISPLACEMENT	. . . . . . D_ commands for displacements
Model verification	Meshing > ELEMENTS Analysis	. . . ECHECK DATA_CHECK, R_CHECK
Specifying analysis options	Analysis > FREQUENCY/BUCKLING	. . . A_BUCKLING
Specifying output options	Analysis > OUTPUTOPTIONS	. . . PRINT_OPS
Executing linear static analysis	Analysis > FREQUENCY/BUCKLING	. . . R_BUCKLING
Postprocessing	Results	Refer to COSMOS/M User Guide (V.1) for more details.

* Please refer to Appendix B, *Command Summary*, for locating the path to menus from cryptic commands.

Modal Analysis

Figure 4-4. Flow Chart for Modal Analysis in the Basic System



Important Commands for Modal Analysis

The commands tabulated below provide you with information on the input of element groups, material and sectional properties, loads and boundary conditions, analysis specifications, and output specifications.

Table 4-3. Important Commands for Modal Analysis

Function	COSMOS/M Menu	Command
Property definition	Propsets	EGROUP, MPROP, RCONST, PICK_MAT, USER_MAT, PICK_SEC, BMSECDEF, EPROPSET, EPROPCHANGE
Loads (only if In-plane effects are considered)	LoadsBC > STRUCTURAL > DISPLACEMENT > FORCE > PRESSURE	. . . . . . D_ commands for displacements F_ commands for forces P_ commands for pressure
Boundary conditions	LoadsBC > STRUCTURAL > DISPLACEMENT > MASTER DOF > COUPLING > BONDING	. . . . . . D_ commands for displacements MD_ commands for Guyan reduction CP_ commands for coupling dof BOND_ commands for bonding dof
Model verification	Meshing > ELEMENTS Analysis	. . . ECHECK DATA_CHECK, R_CHECK
Specifying analysis options	Analysis > FREQUENCY/BUCKLING	. . . A_FREQUENCY
Specifying output options	Analysis > OUTPUT_OPS	. . . PRINT_OPS
Executing linear static analysis	Analysis > FREQUENCY/BUCKLING	. . . R_FREQUENCY
Postprocessing	Results	Refer to COSMOS/M User Guide (V.1) for more details.

* Please refer to Appendix B, *Command Summary*, for locating the path to menus from cryptic commands.

Verification of Model Input Data

One of the difficulties you may come across in the solution of finite element problems is to avoid errors in model input data. Some of the errors can be detected by plotting the model in various positions, listing the element connectivities, listing material and sectional properties, plotting or listing loads and constraints, and many other on-line tools. For small problems, it is often easier to perform these checks to see if all required input data have been properly generated and defined. However, you may still miss some errors that are not easily identifiable. For these types of situations and also for larger problems, it is plausible to perform model checks in an automated environment. COSMOS/M provides powerful tools to verify the robustness and validity of the finite model you build within GEOSTAR automatically. The table below shows the commands you can use in model verification and their functions.

Table 4-4. Commands for Model Input Verification

Command	Menu	Function
ECHECK	Meshing > ELEMENTS	Checks the aspect ratios of specified elements. The command automatically deletes the degenerate elements from the model. The command also checks the element connectivities.
DATA_CHECK	Analysis	Checks if an element group, material property set, and a real (section) constant have been defined for each element in the model.
R_CHECK	Analysis	Performs rigorous checks on the model for validity and completeness for the specified type of analysis.

As you can notice from the above table, the **Data Check** command is a subset of the **Run Check** command. Even though the **Run Check** command identifies elements with bad geometry, the deletion of degenerate elements is performed by the **ECHECK** (Meshing > ELEMENTS > **Check Element**) command.

You are ***strongly*** recommended to use the **Run Check** command and apply any corrections to the finite element model before performing any analysis.

Note that the **Run Check** command is a general model verification tool. You may still find some errors that are *not* trapped by the use of this command. In most cases, the diagnostic messages either printed on the screen or written to the output file provide further information as to the nature of error and its remedies.




5 Element Library

Introduction

The Basic System features an extensive element library to suit your finite element modeling and analysis requirements for all types of practical problems. These elements model behavior of 1D, 2D, and 3D problems in linear static, buckling, and natural frequency and mode shape computations. Some of the elements are available as adaptive finite elements for application in the H-, P-, and HP-methods of analysis.

A detailed description of all elements is provided in the COSMOS/M User Guide (V. 1). In this chapter, a brief description of the available elements and their modeling and analysis options are presented.

The second part of this manual, *Verification Problems*, provides a listing of verification problems by element type. You may review these examples to understand the application of different types of elements available in the Basic FEA System.




Basic System Element Library

The commands for element group definition and manipulation of associated element properties are shown in the table below.

Table 5-1. Commands for Element Group Definition, Modification, and Listing

EGROUP (Propsets > Element Group)	
Defines element groups and the associated element analysis options.	The *maximum* number of element groups permitted in a model is 5000.
EPROPSET (Propsets > New Property Set)	
Assigns *existing* element group, material property, and real constant groups as well as element coordinate system to newly created elements.	—
EPROPCHANGE (Propsets > Change El-Prop)	
Changes the association between element groups, real constants sets, and material property sets.	—
EGLIST (Edit > LIST > Element Groups)	
Lists specified element groups and the associated element analysis options.	The on-screen listing can be *piped* to a text file if desired, using the Control > MISCELLANEOUS > **List Log** command. See Chapter 5 for more discussion on defining and modifying element groups.
EGDEL (Edit > DELETE > Element Groups)	
Deletes specified element groups and the associated element analysis options.	—

The following pages show pictorial representations of all elements available in the Basic System. COSMOS/M User Guide (V. 1) presents a detailed description of all elements in Chapter 4, *Element Library.*

✍ Unless otherwise specified, all commands referred to in this chapter are found in the Propsets and Edit menus.

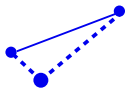
Elements for Linear Static Stress, Buckling, and Modal Analyses

2-D Truss/Spar
Element: TRUSS2D
Nodes: 2

3-D Truss/Spar
Element: TRUSS3D
Nodes: 2

2-D Beam
Element: BEAM2D
Nodes: 2

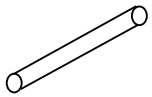
3-D Beam
Element: BEAM3D
Nodes: 2 or 3



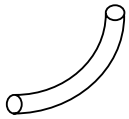
Rigid Bar
Element: RBAR
Nodes: 2

Linear Spring
Element: SPRING
Nodes: 2

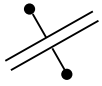
Straight Pipe
Element: PIPE
Nodes: 2



Elbow (Curved Pipe)
Element: ELBOW
Nodes: 3

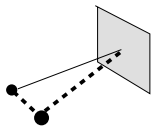


Linear Gap
Element: GAP
Nodes: 2

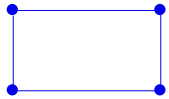


Concentrated Mass
Element: MASS
Nodes: 1

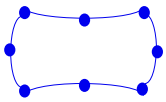
Boundary Element
Element: BOUND
Nodes: 2



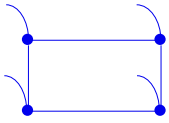
4-node Plane Quadrilateral
Element: PLANE2D
Nodes: 4



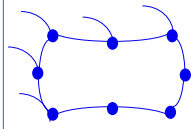
8-node Plane Quadrilateral
Element: PLANE2D
Nodes: 8



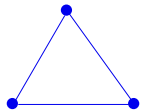
4-node Axisymmetric Quadrilateral
Element: PLANE2D
Nodes: 4



8-node Axisymmetric Quadrilateral
Element: PLANE2D
Nodes: 8

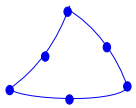


3-node Plane Triangle
Element: TRIANG
Nodes: 3

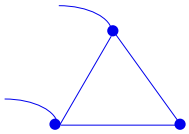


Elements for Linear Static Stress, Buckling, and Modal Analyses (Continued)

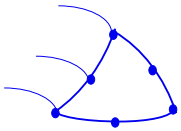
6-node Plane Triangle
Element: TRIANG
Nodes: 6



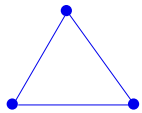
3-node Axisymmetric Triangle
Element: TRIANG
Nodes: 3



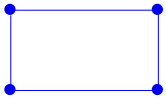
6-node Axisymmetric Triangle
Element: TRIANG
Nodes: 6



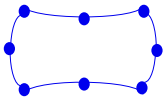
3-Node Thin Shell
Element: SHELL3
Nodes: 3



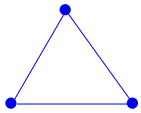
4-Node Thin Shell
Element: SHELL4
Nodes: 4



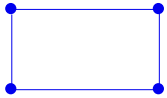
8/9-Node Thin Shell
Element: SHELL9
Nodes: 8/9



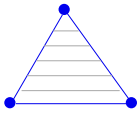
3-Node Thick Shell
Element: SHELL3T
Nodes: 3



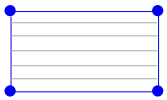
4-Node Thick Shell
Element: SHELL4T
Nodes: 4



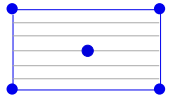
3-Node Composite Shell
Element: SHELL3L
Nodes: 3



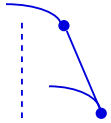
4-Node Composite Shell
Element: SHELL4L
Nodes: 4



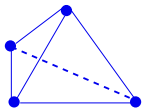
8/9-Node Composite Shell
Element: SHELL9L
Nodes: 8/9



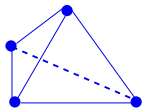
2-Node Axisymmetric Shell
Element: SHELLAX
Nodes: 2



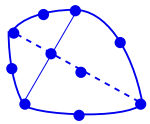
4-Node Tetrahedral Solid
Element: TETRA4
Nodes: 4



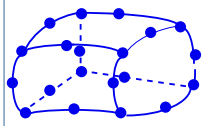
4-Node Tetrahedral Solid with Rotational DOFs
Element: TETRA4R
Nodes: 4



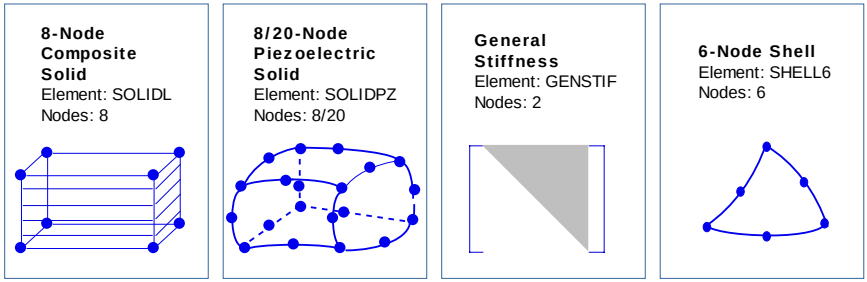
10-Node Tetrahedral Solid
Element: TETRA10
Nodes: 10



8/20-Node Solid
Element: SOLID
Nodes: 8/20



Elements for Linear Static Stress, Buckling, and Modal Analyses (Concluded)



Definition of Element Groups and Related Options

The **EGROUP** (Propsets > **Element Group**) command is used to define the element group number and the element type, and the modeling and analysis options associated with the specified element. A *maximum* of 20 element groups can be defined in your finite element model.

The options *"element group"* is the element group number, and *"element name"* is the valid element name. Table 5-2 shows all valid element names in the **Element Group** command. The elements applicable in the Basic FEA System are also indicated.

The command **EGLIST** (Edit > LIST > **Element Groups**) can be used to list the data defined by the **Element Group** command on the screen. By default, this information is also written to the output file after an analysis has been performed. The command **EGDEL** (Edit > DELETE > **Element Groups**) can be used to delete an existing element group and the associated element options.

Table 5-2. Valid Element Names for the Element Group Command

Element Name	Description	Basic FEA System
BEAM2D	Two dimensional elastic beam element	•
BEAM3D	Three dimensional elastic beam element	•
BOUND	Boundary element	•
BUOY	1-node spherical buoyant element (usually used with IMPIPE element)	N/A
CLINK	Convection link	N/A
ELBOW	Elastic curved pipe (elbow) element	•
ELINK	Electrical link	N/A
FLOW2D	Two dimensional fluid flow element	N/A
FLOW3D	Three dimensional fluid flow element	N/A
GAP	Gap element	•
GENSTIF	General stiffness element	•
HLINK	Hydraulic link	•
IMPIPE	2-node immersed pipe or cable element	N/A
MAG2D	2D magnetic element	N/A
MAG3D	3D magnetic element	N/A
MASS	General mass element	•
PIPE	Elastic Straight pipe element	•
PLANE2D	4- to 8-node plane and axisymmetric element	•
RBAR	Rigid bar element	•
RLINK	Radiation link	N/A
SHELL3	3-node thin shell element	•
SHELL3L	Multi-layer 3-node shell/plate element	•
SHELL3T	3-node thick shell element	•
SHELL4	4-node thin shell element	•
SHELL6	6-node thin shell element	•
SHELL6T	6-node thick shell element	•
SHELL4L	Multi-layer 4-node shell/plate element	•

Table 5-2. Valid Element Names for the Element Group Command (Concluded)

Element Name	Description	Basic FEA System
SHELL4T	4-node thick shell element	•
SHELL9	9/8-node shell element	•
SHELL9L	Composite 9/8-node shell element	•
SHELLAX	Axisymmetric shell element	•
SOLID	8- to 20-node 3D solid element	•
SOLIDL	Composite 8-node solid element	•
SOLIDPZ	8- to 20-node 3D solid piezoelectric element	•
SPRING	Spring element	•
TETRA10	10-node tetrahedral solid element	•
TETRA4	4-node tetrahedral solid element	•
TETRA4R	4-node tetrahedral solid element with translational and rotational dof	•
TRIANG	3- to 9-node triangular plane and axisymmetric element	•
TRUSS2D	Two dimensional truss/spar element	•
TRUSS3D	Three dimensional truss/spar element	•

Every element in the foregoing table has different analysis and modeling options (maximum of seven entries), designated as OP1, …, OP7. When you execute the **Element Group**-command, you are prompted for the physical interpretation for the options depending on the selected element. Some of the arguments in the option field are unused for analysis in the Basic System.

Table 5-3 in the following pages shows a summary of the analysis and modeling options in the Basic System. Table 5-4 shows the output coordinate systems for all elements. Refer to Chapter 4, *Element Library*, of the COSMOS/M User Guide (V. 1) for more information on elements.




Table 5-3. Modeling and Analysis Options for Elements

Element	Element Options						
	OP1	OP2	OP3	OP4	OP5	OP6	OP7
TRUSS2D	*unused*	*unused*	*unused*	*unused*	default	default	default
TRUSS3D	*unused*	*unused*	*unused*	*unused*	default	default	default
BEAM2D	*unused*	*unused*	*unused*	default	default	default	*unused*
BEAM3D	Section Type 0=symmetric 1=un-symm. 2=sym. tapered	*unused*	*unused*	default	default	default	*unused*
RBAR	none						
SPRING	Spring Type 0=axial spring 1=rotat. spring 2=both	*default*	*unused*	*unused*	default	default	*unused*
PIPE	none						
ELBOW	none						
GAP	Surface Update 0=fixed 1=movable	Friction 0=no friction 1=friction, no sliding 2=friction, sliding	default	default	default	Cmprsv Gap Msmnt 0=user defined 1=auto	default
MASS	none						
BOUND	Boundary Type 0=axial spring 1=rotat. spring 2=both	*unused*	*unused*	*unused*	*unused*	*unused*	*unused*
PLANE2D	Element type 0=struct./therm. 1=fluid	Quadrature 0=Red. Int. 1=QM6 2=Full Int.	Prob. Type 0=Pl. Stress 1=Axisymm. 2=Pl. Strain 3=Axisymm.	Str. Output 0=global CS 1=ECS	default	default	default
TRIANG	*unused*	*unused*	Prob. Type 0=Pl. Stress 1=Axisymm. 2=Pl. Strain	Str. Output 0=global CS 1=ECS	*unused*	*unused*	*unused*

Table 5-3. Modeling and Analysis Options for Elements Continued)

Element	Element Options						
	OP1	OP2	OP3	OP4	OP5	OP6	OP7
SHELL3	*unused*	Ana. Option 0=shell 1=membrane 2=shear panel	Stress Print 0=elem. center 1=+nodal force 2=+nodal str.	Nodal Stress Dir. 0=global CS 1=local CS	default	default	*unused*
SHELL4	Element Type 0=QUAD2 1=QUAD4 2=QUAD	Ana. Option 0=shell 1=membrane 2=shear panel	Stress Print 0=elem. center 1=+nodal force 2=+nodal str.	Nodal Stress Dir. 0=global CS 1=local CS	default	default	*unused*
SHELL6	*unused*	Integ. type 0=Red. 1=full	*unused*	Nodal Stress Dir. 0=global CS 1=local CS	*unused*	*unused*	*unused*
SHELL3T	*unused*	Ana. Option 0=shell 1=membrane 2=shear panel	Stress Print 0=elem. center 1=+nodal force 2=+nodal str.	Nodal Stress Dir. 0=global CS 1=local CS	default	default	*unused*
SHELL4T	Element Type 0=QUAD2 1=QUAD4 2=QUAD	Ana. Option 0=shell 1=membrane 2=shear panel	Stress Print 0=elem. center 1=+nodal force 2=+nodal str.	Nodal Stress Dir. 0=global CS 1=local CS	default	default	*unused*
SHELL6T	*unused*	Ana. Option 0=shell 1=membrane 2=shear panel	Stress Print 0=elem. center 1=+nodal force 2=+nodal str.	Nodal Stress Dir. 0=global CS 1=local CS	default	default	*unused*
SHELL3L	*unused*	No. of Layers	Nodal Str. Dir. 0=global CS 1=local CS 2=material CS	Layer number for plotting stresses	default	default	Input 0=layers 1=sandwich
SHELL4L	Element Type 0=QUAD2 1=QUAD4 2=QUAD	No. of Layers	Nodal Str. Dir. 0=global CS 1=local CS 2=material CS	unused	default	default	Input 0=layers 1=sandwich
SHELL9	No. of Nodes 0=9 nodes 1=8 nodes	Quadrature 0=γ - ϕ method 1=Red. Integr.	*unused*	Nodal Stress Dir. 0=global CS 1=local CS	*unused*	*unused*	*unused*

Table 5-3. Modeling and Analysis Options for Elements (Concluded

Element	Element Options						
	OP1	OP2	OP3	OP4	OP5	OP6	OP7
SHELL9L	No. of Nodes 0=9 nodes 1=8 nodes	Quadrature 0=γ - ϕ method 1=Red. Integr.	No. of Layers	Layer number for plotting stresses	Stress Dir. 0=glob. 1=local	*unused*	*unused*
SHELLAX	none						
TETRA4	N/A	*unused*	*unused*	Str. Output 0=global CS 1=ECS	*unused*	*unused*	*unused*
TETRA4R	N/A	*unused*	*unused*	Str. Output 0=global CS 1=ECS	*unused*	*unused*	*unused*
TETRA10	N/A	*unused*	*unused*	Str. Output 0=global CS 1=ECS	*unused*	*unused*	*unused*
SOLID	Element Type 0=solid 1=fluid	Quadrature 0=red. Int. 1=hybrid 2=full Int.	*unused*	Str. Output 0=global CS 1=ECS	default	default	default
SOLIDL	Quadrature 0=red. Int. 1=hybrid 2=full Int.	No. of Layers	Nodal Str. Dir. 0=global CS 1=local CS 2=material CS	Layer number for plotting stresses	*unused*	*unused*	*unused*
SOLIDPZ	Integration Type 1=hybrid 2=full	*unused*	Nodal Str. Dir. 0=global CS 1=local CS 2=material CS	Print Str. 0=Top & Bottom.	*unused*	*unused*	*unused*
GENSTIF	none						

Table 5-4. Element Output Coordinate Systems

Element Name	Element Local Coordinate System (ECS=-1, or 1)				Global (ECS=0)
	Default	Cartesian	Cylindrical	Spherical	
TRUSS2D	-1	0	1	2	0
TRUSS3D	-1	0	1	2	0
BEAM2D	-1	0	1	2	0
BEAM3D	-1	0	1	2	0
RBAR	-1	0	1	2	0
SPRING	-1	0	1	2	0
PIPE	-1	0	1	2	0
ELBOW	-1	0	1	2	0
GAP	-1	0	1	2	0
MASS	-1	0	1	2	0
BOUND	-1	0	1	2	0
PLANE2D 4-NODE	-1	0	1	2	0
PLANE2D 8-NODE	-1	0	1	2	0
PLANE2D 4-NODE (Axi.)	-1	0	1	2	0
PLANE2D 8-NODE (Axi.)	-1	0	1	2	0
TRIANG 3-NODE	-1	0	1	2	0
TRIANG 6-NODE	-1	0	1	2	0
TRIANG 3-NODE (Axi.)	-1	0	1	2	0
TRIANG 6-NODE (Axi.)	-1	0	1	2	0
SHELL3	-1	0	1	2	0
SHELL 6	-1	0	1	2	0
SHELL4	-1	0	1	2	0
SHELL3T	-1	0	1	2	0
SHELL4T	-1	0	1	2	0
SHELL3L	-1	0	1	2	0
SHELL4L	-1	0	1	2	0
SHELL9	-1	0	1	2	0
SHELL9L	-1	0	1	2	0
SHELLAX	-1	0	1	2	0
TETRA4	-1	0	1	2	0
TETRA4R	-1	0	1	2	0
TETRA10	-1	0	1	2	0
SOLID	-1	0	1	2	0
SOLIDL	-1	0	1	2	0
SOLIDPZ	-1	0	1	2	0
GENSTIF	-1	0	1	2	0




Property Definitions for Elements

The material and sectional properties for elements are defined using commands in the Propsets menu. The table below summarizes the functions of commands in this menu.

Table 5-5. Commands for Property Definitions (Propsets and Edit Menus)

Command	Description	Remarks
MPROP (Propsets > Material Property)	Defines a material property set and the property values for valid material names (see Table 6-2) in this set.	The maximum number of material property sets allowed in a model is 90.
PICK_MAT (Propsets > Pick Material Lib)	Selects a material from the built-in material library.	There are 14 materials in the built-in material library.
USER_MAT (Propsets > User Material Lib)	Selects a material from the user-defined material library.	The format of the user-defined library must be the same as that of the built-in library, and should be named USERMAT.LIB in the COSMOS/M directory.
R_MATLIB (Propsets > Material Browser)	Runs the add-on material browser to select materials from an extensive library that includes temperature and stress-strain curves.	It is required to purchase the material browser before using this utility.
MPLIST (Edit > LIST > Material Props)	Lists material properties defined for the specified sets on the screen.	You can also route the on-screen listing of material properties to a file.
MPDEL (Edit > DELETE > Material Props)	Deletes material properties for the specified set(s).	
RCONST (Propsets > Real Constant)	Defines a section property set and the property values applicable (see Table 6-11) for the type of element in this set.	The maximum number of real constant sets is 1000. The element group must be selected prior to executing this command.
PICK_SEC (Propsets > AISC Sect. Table)	Selects a specified shape of AISC section from the built-in section library and the corresponding section properties.	See AISC handbook for more information on standard steel sections.
RCLIST (Edit > LIST > Real Constants)	Lists section and other physical properties defined for specified sets on the screen.	The on-screen listing can be *piped* to a text file if desired, using the **List Log** command.
RCDEL (Edit > DELETE > Real Constants)	Deletes section and other physical properties for the specified sets.	

The next chapter provides an in-depth coverage of property definitions for elements in the Basic System. The table below provides a summary of available material and sectional properties in the Basic System. Table 5-7 lists the available built-in material and section library features for elements.

Table 5-6. Summary of Properties for Elements in the Basic FEA System

COSMOS/M Element Name	Material Properties					Section Constant	Mass		Element Loads		
	Iso	Orth	Anis	Th	Pz		Lu	Dis	Me	Th	Gr
TRUSS2D	•			•		2	•	•		•	•
TRUSS3D	•			•		2	•	•		•	•
BEAM2D	•			•		7	•	•	•	•	•
BEAM3D	•			•		14 to 27	•	•	•	•	•
RBAR	N/A	N/A	N/A	N/A	N/A	1	•	•	N/A	N/A	N/A
SPRING	N/A	N/A	N/A	N/A	N/A	2	•	•	N/A	N/A	N/A
PIPE	•			•		3	•	•	•	•	•
ELBOW	•			•		4	•	•	•	•	•
GAP	N/A	N/A	N/A	N/A	N/A	2	N/A	N/A			
MASS	N/A	N/A	N/A	N/A	N/A	7	•		N/A	N/A	N/A
BOUND	N/A	N/A	N/A	N/A	N/A	2	N/A	N/A			
PLANE2D 4-NODE	•	•		•		2	•	•	•	•	•
PLANE2D 8-NODE	•	•		•		2	•	•	•	•	•
PLANE2D 4-NODE (Axi.)	•	•		•		1	•	•	•	•	•
PLANE2D 8-NODE (Axi.)	•	•		•		1	•	•	•	•	•
TRIANG 3-NODE	•	•		•		2	•	•	•	•	•
TRIANG 6-NODE	•	•		•		2	•	•	•	•	•
TRIANG 3-NODE (Axi.)	•	•		•		1	•	•	•	•	•
TRIANG 6-NODE (Axi.)	•	•		•		1	•	•	•	•	•
SHELL3	•			•		3	•	•	•	•	•
SHELL4	•			•		3	•	•	•	•	•
SHELL 6	•	•		•		2	•	•	•	•	•
SHELL3T	•			•		3	•	•	•	•	•
SHELL4T	•			•		3	•	•	•	•	•
SHELL3L	•	•	•	•		2+3NL*	•	•	•	•	•
SHELL4L	•	•	•	•		2+3NL*	•	•	•	•	•
SHELL9	•			•		1	•	•	•	•	•
SHELL9L	•	•		•		2+3NL*	•	•	•	•	•
SHELLAX	•			•		1	•	•	•		•
TETRA4	•	•		•		N/A	•	•	•	•	•
TETRA4R	•	•	•	•		N/A	•	•	•	•	•
TETRA10	•	•	•	•		N/A	•	•	•	•	•
SOLID	•	•	•	•		9**	•	•	•	•	•
SOLIDL	•	•		•		2+6NL*	•	•	•	•	•
SOLIDPZ	•	•	•	N/A	•	N/A	•	•			
GENSTIF	•	•	•	N/A	•	N/A	•	•	N/A	N/A	N/A

* depends on the number of layers (NL)

** for defining orthotropic material axis orientation only

Table 5-7. Material and Section Libraries for Elements in the Basic FEA System

Element Name	Material Library		Section Library	
	Built-In	User-Defined	COSMOS/M	AISC
TRUSS2D	•	•		
TRUSS3D	•	•		
BEAM2D	•	•	•	•
BEAM3D	•	•	•	•
RBAR	N/A	N/A	N/A	N/A
SPRING	N/A	N/A		
PIPE	•	•		
ELBOW	•	•		
GAP	N/A	N/A	N/A	N/A
MASS	N/A	N/A	N/A	N/A
BOUND	N/A	N/A	N/A	N/A
PLANE2D 4-NODE	•	•		
PLANE2D 8-NODE	•	•		
PLANE2D 4-NODE (Axi.)	•	•		
PLANE2D 8-NODE (Axi.)	•	•		
TRIANG 3-NODE	•	•		
TRIANG 6-NODE	•	•		
TRIANG 3-NODE (Axi.)	•	•		
TRIANG 6-NODE (Axi.)	•	•		
SHELL3	•	•		
SHELL4	•	•		
SHELL3T	•	•		
SHELL 6	•	•		
SHELL4T	•	•		
SHELL3L	*	*		
SHELL4L	*	*		
SHELL9	•	•		
SHELL9L	*	*		
SHELLAX	•	•		
TETRA4	•	•		
TETRA4R	•	•		
TETRA10	•	•		
SOLID	•	•		
SOLIDL	*	*		
SOLIDPZ	N/A	N/A		
GENSTIF	N/A	N/A		

* A different material can be assigned to each layer (see Chapter 6)

Elements for Adaptive FEA

In the Basic System, an automated approach is available for mesh refinement based on reducing errors due to mesh discretization. The H-method of the finite element method refers to a convergence based on progressive reduction of element size whereas the P-method is refers to the refinement approach in which the polynomial degree is adaptively refined. The combined approach is called the HP-method.

The table below summarizes the adaptive method and the applicable elements in the Basic System.

Table 5-8. Elements for Adaptive Analysis in the Basic System

Adaptive Method	Available Elements
H-method	TRIANG, SHELL3, SHELL3T, TETRA4, TETRA4R, TETRA10
P-method	6-node TRIANG, 8-node PLANE2D, SHELL9L,TETRA10
HP-method	6-node TRIANG, TETRA10

Please refer to COSMOS/M User Guide (V. 1) for more information on adaptive finite element analysis.







6 Material, Sectional and Other Physical Properties

Introduction

In finite element analysis, the elements provide the spatial approximation to the model geometry, whereas the mathematical models for material properties provide the physical representation. It is well known that the accuracy of finite element analysis depends on the accuracy with which these idealizations are carried out, in addition to various other factors. Accurate description of material properties is therefore an important aspect of finite element modeling. A comprehensive finite element solver should provide all required capabilities for accurately describing the physical nature of the material in your model. COSMOS/M meets this goal by providing elaborate and extensive yet tractable capabilities for describing the material, sectional, and other types of physical properties in your finite element model.

In the Basic FEA System, the material property definitions include elastic, thermoelastic, mass, and piezoelectric properties. The mass density is required when the analysis type needs mass information. For example, in natural frequency calculations and in elastostatic analysis with inertial loading, mass density is required to be input. The material property definitions also include the allowable stresses for computing the failure indices in layered composites. The sectional and other physical property definitions include cross sectional area, moment of inertia, depth or thickness, perimeter, temperature gradients across the thickness, end release conditions, and other information for computing the element matrices. They also include the definition of spring constants and friction coefficients.

In order to numerically evaluate the behavior of finite element models, the material properties input are applied in various mathematical models, otherwise known as *Constitutive Relations*. Hooke's law which represents a linear relation between stress and strain is a simple form of constitutive relation. This chapter presents a detailed description of the various material properties required for analysis in the Basic FEA System and the corresponding constitutive relations. The commands and procedures to accomplish these along with the definition of sectional and other physical properties are also presented. The descriptions are supplemented with many illustrations and examples to make your understanding clear and comprehensive.

Property Definitions in COSMOS/M

The definition of *element type* to be used in the analysis as well as the associated *material* and *sectional properties* are specified using the Propsets, Control > MEASURE, Edit > LIST, and Edit > DELETE menus in COSMOS/M. The table below outlines the function of all commands in this menu.

Table 6-1. Functions of Commands in the Propsets, Control > MEASURE, Edit > LIST, and Edit > DELETE Menus

Command	Description	Remarks
MPROP (Propsets > Material Property)	Defines a material property set and the property values for valid material names (see Table 6-2) in this set.	The maximum number of material property sets allowed in a model is 90.
PICK_MAT (Propsets > Pick Material Lib)	Selects a material from the built-in library.	There are 45 materials in the built-in library.
USER_MAT (Propsets > User Material Lib)	Selects a material from the user-defined material library.	The format of the user-defined library must be the same as that of the built-in library.
MPLIST (Edit > LIST > Material Props)	Lists material properties defined for the specified sets on the screen.	You can also route the on-screen listing of material properties to a file using LISTLOG.
MPDEL (Edit > DELETE> Material Props)	Deletes material properties for the specified set(s).	
RCONST (Propsets > Real Constant)	Defines a section property set and the property values applicable (see Table 6-2) for the type of element in this set.	The maximum number of real constant sets is 5000. The element group *must* be selected prior to executing this command.

Table 6-1. Functions of Commands in the Propsets, Control > MEASURE, Edit > LIST, and Edit > DELETE Menus (Concluded)

Command	Description	Remarks
RCONST (Propsets > Real Constant)	Defines a section property set and the property values applicable (see Table 6-2) for the type of element in this set.	The maximum number of real constant sets is 5000. The element group *must* be selected prior to executing this command.
PICK_SEC (Propsets > AISC Sect Table)	Selects a specified shape of AISC section from the built-in section library and the corresponding section properties.	See AISC handbook for more information on standard steel sections.
RCLIST (Edit > LIST > Real Constants)	Lists section and other physical properties defined for specified sets on the screen.	The on-screen listing can be *piped* to a text file if desired, using the LISTLOG command.
RCDEL (Edit > DELETE> Real Constants)	Deletes section and other physical properties for the specified sets.	
MASSPROP (Control > MEASURE > Find Mass Property)	Computes length, area, volume, mass, center of gravity, moment of inertia, radius of gyration, direction cosines, etc., for selected elements and displays the results on the screen.	The on-screen listing can be *piped* to a text file if desired, using the LISTLOG command.
EGROUP (Propsets > Element Group)	Defines element groups and the associated element analysis options.	The maximum number of element groups permitted in a model is 20.
EPROPSET (Propsets > New Property Set)	Assigns *existing* element group, material property, and real constant groups as well as element coordinate system to newly created elements.	
EPROPCHANGE (Propsets > Change El-Prop)	Changes the association between element groups, real constants sets, and material property sets.	The active element group, real constant or material property sets can be listed using the STATUS1 command.
EGLIST (Edit > LIST > Element Groups)	Lists specified element groups and the associated element analysis options.	See Chapter 5 for more discussion on defining and modifying element groups.
EGDEL (Edit > DELETE> Element Groups)	Deletes specified element groups and the associated element analysis options.	
BMSECDEF (Propsets > Beam Section)	Selects a standard beam cross section from the built-in section library.	There are 10 standard sections in addition to a user-defined section of arbitrary shape.
BMSECLIST (Edit > LIST> Beam Section)	Lists the selected standard beam cross section.	




This chapter focuses on presenting you with in-depth information on defining the material, sectional, and other physical properties of your finite element model. Related theoretical concepts and mathematical models are briefly explained (for a more detailed theoretical discussion, please refer to the COSMOS/M Theoretical Manual). However, you will learn more in detail about how to use these mathematical models in COSMOS/M and the required input for proper usage. This chapter also discusses the procedure to compute mass and inertia properties of structures using the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command.

In order to preserve the clarity of presentation, the element groups definitions, corresponding element option selections, and the modification of these definitions are discussed in Chapter 5, *Element Library.*

✍ Unless otherwise specified, all commands referred to in this chapter are found in the Propsets, Control > MEASURE, Edit > LIST, and Edit > DELETE menus.

Almost all commands referenced in this manual are the cryptic ones which are *actually* parsed and interpreted by the program. Enhanced (or English-like) commands you also see on the screen *only* serve the purpose of increasing the readability and user-friendliness of the graphical interface.

A Note About Units

In COSMOS/M, you can adopt any system of units, including standard and non-standard systems. However, units must be consistent and you should interpret the results accordingly. Appendix A presents a detailed description of consistent units applicable to analyses in the Basic FEA System. This appendix also features conversion factors from one system of units to another, and other useful information.

If you are making use of the built-in material library in COSMOS/M (using **PICK_MAT** (Propsets > **Pick Material Lib**) or **USER_MAT** (Propsets > **User Material Lib**) commands), a word of caution about units is in order. The material properties in these material libraries are available in FPS, MKS, and SI systems. However, the property values stored are *not* literal versions of units implied in these systems. The *only* exception is the SI system.

- Even though the FPS system implies foot as the unit of length, the built-in material libraries use inch. This was done to make it easier for the practicing engineers and scientists who can relate to the prevailing industry standards.

- On the same token, if you choose the MKS system, the built-in material libraries use centimeter as the unit of length instead of meter. You need to keep this in mind when building a finite element model.

Material Properties in COSMOS/M

All material properties in COSMOS/M are defined using the command **MPROP** (Propsets > **Material Property**). This command assigns a material set number under which you can define more than *fifty* different material properties. The defined material set number can be related to one or more element group numbers. Table 6-2 shows the valid material names you can use for defining the required material properties. Some of the property names in this table are *not* applicable in the Basic FEA System.

All valid material names from Table 6-2 can be assigned to one set. Each material name may have one or more (up to 21) property values to be input. The commands **PICK_MAT** (Propsets > **Pick Material Lib**) and **USER_MAT** (Propsets > **User Material Lib**) can also be used to define a material set and select properties from the built-in or your own material libraries for that set. The properties selected from the libraries can be further appended by executing the **MPROP** (**Material Property**) command.

The command **MPDEL** (Edit > DELETE > **Material Props**) can be used to delete a material property set. If you *only* need to modify or delete a particular property in a material set, you need to re-execute the **MPROP** (**Material Property**) command and specify either zero or any other required value for that property name. For more information on modification of material property definitions, please refer to the section, *Modification, Display, and Output of Material Property Definitions*.

The command **MPLIST** (Edit > LIST > **Material Props**) can be used to list all material properties associated with a set on the screen. If you request for a detailed output using the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command, the material property data will also be written to the output file (jobname.OUT) after an analysis is performed.




To define material properties, execute the **MPROP** (**Material Property**) command and specify the required property names from Table 6-2. You will be again prompted to specify another property name (and the corresponding property value if you respond to the prompt) for a *maximum* of 21 times. If you need to define more than 21 material properties under the same material set, you need to re-execute the **MPROP** (**Material Property**) command. The significance of number 21 is that it represents the maximum number of elastic coefficients to be input for anisotropic materials. Most other materials with isotropic or orthotropic elastic and thermal properties can be adequately described with fewer constants. The **MPROP** (**Material Property**) command is illustrated below:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **EX: Elasticity modulus in X mat. dir.**
Property value > **20E6**

Table 6-2. Property Names in COSMOS/M

Property Name	Description	Basic System	Units
ALPH1, ALPH2, ALPH3, ALPH4	First, second, third and fourth power coefficients of Ogden material model	N/A	
ALPX, ALPY, ALPZ	Coefficient of thermal expansion in material X-, Y-, and Z-directions	•	Strain/Temp
BETA	Coefficient of volume expansion	N/A	
C	Specific heat	N/A	Heat/ (Mass•Degree)
COHESN	Cohesion strength	N/A	Force/Area
CREEPC	Creep constants for the classical creep law (three constants are needed)	N/A	*none*
CREEPX	Creep constants for the exponential creep law (seven constants are needed)	N/A	*none*
DAMP	Material Damping coefficient	N/A	*none*
DC_{ij}	Coefficients of the dielectric matrix (six constants are needed)	•	
DENS	Mass density	•	Mass/Volume
ECONX, ECONY, ECONZ	Electric conductivity in the global Cartesian X-, Y-, and Z-directions	N/A	Current/Potential
EMIS	Emissivity (Radiation)	N/A	*none*
ETAN	Tangent elastic modulus	N/A	Force/Area
EX, EY, EZ	Elasticity modulus in X-, Y-, and Z-directions	•	*same as above*

Table 6-2. Property Names in COSMOS/M (Continued)

Property Name	Description	Basic System	Units
FRCANG	Friction angle	N/A	Degrees
G1 through G8	Shear relaxation moduli (for viscoelasticity)	•	
GAMMA	Ratio of specific heats	N/A	
GXY, GXZ, GYZ	Shear modulus, X-Y, X-Z, and Y-Z directions	•	Force/Area
HC	Convection film coefficient	N/A	Heat/(Time•Area•Degree)
K1 through K8	Bulk relaxation moduli (for viscoelasticity)	N/A	
KX, KY, KZ	Thermal conductivity in the global Cartesian X-, Y-, and Z-directions	N/A	Heat•Length/(Time•Area•Degree)
MC_{ij}	Anisotropic elastic moduli (twenty one constants are needed)		Force/Area *or* Area/Force
MOONEY_A, MOONEY_B, MOONEY_C, MOONEY_D, MOONEY_E, MOONEY_F	First, second, third, fourth, fifth, and sixth material constants for Mooney-Rivlin hyperelastic material model	N/A	Force/Area
MPERM	Magnetic permeability	N/A	Inductance/Length
MU1 through MU4	First, second, third and fourth constants for Ogden material model	N/A	
NUXY	Poisson's ratio, strain in the second material direction due to unit strain in the first material direction	•	*none*
NUXZ	Poisson's ratio, strain in the third material direction due to unit strain in the first material direction	•	*none*
NUYZ	Poisson's ratio, strain in the third material direction due to unit strain in the second material direction	•	*none*
PC_{ij}	Coefficients of the piezoelectric material matrix (eighteen constants are needed)	•	
PERMIT	Permittivity (dielectric constant)	N/A	Farads/Length
PMAGR, PMAGT	Coercivity of a permanent magnet in the radial and tangential directions of the global cylindrical system	N/A	Amperes/Length
PMAGX, PMAGY, PMAGZ	Coercivity of a permanent magnet in the global Cartesian X-, Y-, and Z-direction	N/A	Amperes/Length

Table 6-2. Property Names in COSMOS/M (Concluded)

Property Name	Description	Basic System	Units
REFTMP	Glassy transition temperature for viscoelasticity	N/A	
SIGXC	Compressive strength in material X-direction	•	Force/Area
SIGXT	Tensile strength in material X-direction	•	*same as above*
SIGXY	Shear strength in material X-Y plane	•	*same as above*
SIGYC	Compressive strength in material Y-direction	•	*same as above*
SIGYLD	Yield stress	N/A	*same as above*
SIGYT	Tensile strength in material Y-direction	•	*same as above*
TAUG1 through TAUG8	Time values associated with shear relaxation moduli G1 through G8 (used in viscoelasticity)	N/A	
TAUK1 through TAUK8	Time values associated with bulk relaxation moduli K1 through K8 (used in viscoelasticity)	N/A	
VC1, VC2	Constants for the William-Landel-Ferry equation used in viscoelasticity	N/A	
VISC	Dynamic Viscosity	N/A	Force•Time/Area

The types of material descriptions applicable in linear static stress analysis encompass *elastic*, *thermoelastic*, *piezoelectric*, and *mass* properties. Elastic properties include the various elastic moduli and Poison's ratios. If the elastic properties are temperature sensitive, then the material description should use thermoelastic properties which includes the coefficient of thermal expansion as well as some form of dependencies of the elastic moduli and Poison's ratios on temperature. Piezoelectric material description includes the elastic, dielectric, and piezoelectric properties required to be input in *electroelastic* analysis. Mass property includes the density of the material which can also be made temperature dependent.

The material descriptions mentioned above can be applied to all available elements in the Basic FEA System, with the *only* exception of piezoelectric description which is applicable to the SOLIDPZ element. The types of material descriptions available for all elements in the Basic FEA System are shown in Table 6-3. Note that the mass property is applicable to all elements in this table with the *exception* of BOUND, GAP, GENSTIF, RBAR, and SPRING elements.

The description of elastic property covers the definition of isotropic, orthotropic, and anisotropic properties and the procedure to input them in COSMOS/M, followed by a section for the description of thermoelastic property. Properties for layered composites as well as the available failure criteria are discussed in a separate section, followed by a section for the dielectric and piezoelectric matrices and their input for electroelastic analysis. There is also a section for mass and damping properties. The use of the built-in material library is fully documented in another section.

Table 6-3. Material Properties for Elements in the Basic System

COSMOS/M Element Name	Linear Elastic Properties			Thermoelastic Property			Piezoelectric Property
	Iso	Ortho	Anis	Iso	Ortho	Anis	
TRUSS2D	•			•			
TRUSS3D	•			•			
BEAM2D	•			•			
BEAM3D	•			•			
RBAR	N/A	N/A	N/A	N/A	N/A	N/A	N/A
SPRING	N/A	N/A	N/A	N/A	N/A	N/A	N/A
PIPE	•			•			
ELBOW	•			•			
GAP	N/A	N/A	N/A	N/A	N/A	N/A	N/A
MASS	N/A	N/A	N/A	N/A	N/A	N/A	N/A
BOUND	N/A	N/A	N/A	N/A	N/A	N/A	N/A
PLANE2D 4-NODE	•	•		•	•		
PLANE2D 8-NODE	•	•		•	•		
PLANE2D 4-NODE (Axi.)	•	•		•	•		
PLANE2D 8-NODE (Axi.)	•	•		•	•		
TRIANG 3-NODE	•	•		•	•		
TRIANG 6-NODE	•	•		•	•		
TRIANG 3-NODE (Axi.)	•	•		•	•		
TRIANG 6-NODE (Axi.)	•	•		•	•		
SHELL3	•			•			
SHELL4	•			•			
SHELL3T	•			•			

Table 6-3. Material Properties for Elements in the Basic System (Concluded)

COSMOS/M Element Name	Linear Elastic Properties			Thermoelastic Property			Piezoelectric Property
	Iso	Ortho	Anis	Iso	Ortho	Anis	
SHELL4T	•			•			
SHELL3L	•	•	•	•	•		
SHELL4L	•	•	•	•	•		
SHELL6	•	•		•			
SHELL9	•			•			
SHELL9L	•	•		•	•		
SHELLAX	•			•			
TETRA4	•	•	•	•		•	
TETRA4R	•	•	•	•		•	
TETRA10	•	•	•	•		•	
SOLID	•	•	•	•	•	•	
SOLIDL	•	•		•	•		
SOLIDPZ	•	•	•	N/A	•	N/A	•
GENSTIF	•	•	•	N/A	N/A	N/A	•
In the above table: **Iso** = isotropic, **Ortho** = orthotropic, **Anis** = anisotropic.							

Elastic Material Properties

The behavior of a material is said to be elastic when its displacement is linearly proportional to the applied load, and returns to its undeformed state when the load is removed. For elastic materials, stress is directly proportional to the strain as indicated below:

$$\{\sigma\} = [D]\ \{\varepsilon - \varepsilon^t\} \qquad \textbf{(6-1)}$$

where [D] is the elastic (isotropic, anisotropic, or orthotropic) material or material stiffness matrix, $\{\sigma\}$ is the total stress vector, $\{\varepsilon\}$ is the total strain vector, and $\{\varepsilon^t\}$ is the thermal strain vector. The above rule is an approximation and is valid as long as the strains are small, for certain types of materials. The inverse form of the above equation can be written as:

$$\{\varepsilon - \varepsilon^t\} = [C]\ \{\sigma\} \qquad \textbf{(6-2)}$$

where [C] represents the elastic material compliance matrix, and it is the inverse of the elastic material matrix [D]. Equations 6-1 and 6-2 are also known as constitutive relations which assume a simple form for linear elastic materials. However, for nonlinear materials (available in NSTAR), the constitutive relations can be quite complex.

In COSMOS/M, for anisotropic materials, you can input the coefficients of either [C] or [D]. However, for isotropic and orthotropic materials, you can only input the coefficients of [D]. The definition of isotropic, anisotropic and orthotropic materials are provided in the following sections.

In order for you to better understand the linear elastic constitutive relations and the corresponding input, the components of displacement, stress, and strain vectors encountered in one-, two-, and three-dimensional elastostatic analysis are shown in the table below:

Table 6-4. Displacement Components, Stress and Strain Vectors in Elastostatic Analysis

Problem Type	Displacement Components	Strain Vector $\{\varepsilon\}^T$	Stress Vector $\{\sigma\}^T$
Bar (Truss)	u	$[\varepsilon_x]$	$[\sigma_x]$
Beam	w	$[\theta_x]$	$[M_x]$
Plane Stress	u,v	$[\varepsilon_x\ \varepsilon_y\ \gamma_{xy}]$	$[\sigma_x\ \sigma_y\ \tau_{xy}]$
Plane Strain	u,v	$[\varepsilon_x\ \varepsilon_y\ \gamma_{xy}]$	$[\sigma_x\ \sigma_y\ \tau_{xy}]$
Axisymmetric	r,θ,z	$[\varepsilon_r\ \varepsilon_\theta\ \gamma_{r\theta}\ \varepsilon_z]$	$[\sigma_r\ \sigma_\theta\ \tau_{r\theta}\ \sigma_z]$
Three-dimensional	u,v,w	$[\varepsilon_x\ \varepsilon_y\ \varepsilon_z\ \gamma_{xy}\ \gamma_{yz}\ \gamma_{zx}]$	$[\sigma_x\ \sigma_y\ \sigma_z\ \tau_{xy}\ \tau_{yz}\ \tau_{zx}]$
Plate Bending	w	$[\theta_x\ \theta_y\ \theta_{xy}]$	$[M_x\ M_y\ M_{xy}]$
Notation	$\varepsilon_x = u_{,x}\quad \varepsilon_y = v_{,y}\quad \gamma_{xy} = u_{,y} + v_{,x}$ $\theta_x = -w_{,xx}\quad \theta_y = w_{,yy}\quad \theta_{xy} = 2w_{,xy}$ where "," denotes partial derivaton, for example: $\varepsilon_x = \partial u/\partial x$		

Isotropic Elastic Properties

If the properties of a material do not vary with direction, then it is said to be isotropic. Isotropic materials therefore have identical elastic modulus, Poisson's ratio, coefficient of thermal expansion, thermal conductivity, etc. in any direction. The term *isothermal* is some times used to denote materials with no preferred directions for coefficients of thermal expansion.

In order to define the isotropic elastic properties, you can input a combination of any *two* of the following coefficients: elastic modulus E_x, Poisson's ratio ν_{xy}, and shear modulus, G_{xy}. The third coefficient *not* input is internally computed by the program using the relation:

$$E = 2G(1 + \nu)$$

The [C] or [D] matrix for an isotropic material contains only two independent coefficients.

The following sections describe the isotropic stress-strain relations in two- and three-dimensions as well as for axisymmetric solids including the effect of thermal strains.

Two Dimensional Isotropic Stress-Strain Relations

For problems with in-plane loading, finite element analysis is frequently performed using plane elements for which two dimensional stress-strain relations are applied. The deformation states in two dimensions can be either plane stress or plane strain, and for either one of these states to prevail, the plane under consideration (normally, the x-y plane) must be a plane of elastic symmetry.

Plane Stress

Plane stress represents a condition that prevails in a flat plate, normally in the x-y plane, loaded only in its own plane and without z-direction restraint so that the z-components of stresses are zero. For an isotropic material, the stress-strain relations with thermal effects in plane stress are as shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T-T_0) \begin{Bmatrix} \alpha \\ \alpha \\ 0 \end{Bmatrix} \right) \quad \textbf{(6-3)}$$

Plane stress elastic constitutive relations are applicable to PLANE2D and TRIANG elements only. After having specified these element names using the **EGROUP** (Propsets > **Element Group**) command, you need to input the plane stress condition under the third option, OP3; see Chapter 5, *Element Library*, for more information.

Plane Strain

Plane strain represents a deformation state in which the z-components of strains are zero. The displacements in the x-y plane are functions of only x and y but not of z. For an isotropic material, the stress-strain relations with thermal effects in plane strain are as shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T-T_0)(1+\nu) \begin{Bmatrix} \alpha \\ \alpha \\ 0 \end{Bmatrix} \right)$$

(6-4)

The z-component of stress (σ_z) does not appear in the above equation even though it is usually not zero. However, it can be calculated once σ_x and σ_y are known.

Plane strain elastic constitutive relations are applicable to PLANE2D and TRIANG elements only. After having specified these element names using the **EGROUP** (Propsets > **Element Group**) command, you need to input the plane strain condition under the third option, OP3; see Chapter 5, *Element Library*, for more information.

Axial Symmetry

In many practical applications, the models are solids of revolution, and are therefore axially symmetric. The behavior of such models is independent of the circumferential coordinate. A two dimensional slice of the model can be used to accurately simulate the three dimensional problem. Axially symmetric solids therefore require a 4 by 4 elastic material matrix, as shown below:

$$\begin{Bmatrix} \sigma_r \\ \sigma_\theta \\ \tau_{r\theta} \\ \sigma_z \end{Bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 & \frac{\nu}{1-\nu} \\ \frac{\nu}{1-\nu} & 1 & 0 & \frac{\nu}{1-\nu} \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} & 0 \\ \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 1 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \gamma_{r\theta} \\ \varepsilon_z \end{Bmatrix} - (T-T_0) \begin{Bmatrix} \alpha \\ \alpha \\ 0 \\ \alpha \end{Bmatrix} \right)$$

(6-5)

The equivalence for stress (or strain) components between axisymmetric solids and general three dimensional solids is as follows: $\sigma_r = \sigma_x$, $\sigma_\theta = \sigma_y$, $\sigma_z = \sigma_z$.

Axisymmetric elastic constitutive relations are applicable to PLANE2D, TRIANG, and SHELLAX elements only. For the axisymmetric shell element, these relations applied by default. But for PLANE2D and TRIANG elements, using the **EGROUP** (Propsets > **Element Group**) command, you need to specify the axisymmetric condition under the third option, OP3; see Chapter 5, *Element Library*, for more information.

Three-Dimensional Isotropic Stress-Strain Relations

The most general form of the isotropic stress-strain relations including thermal effects is shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \\ 0 \\ 0 \end{Bmatrix} \right) \quad \textbf{(6-6)}$$

Three dimensional stress-strains relations are applied by default to all solid elements, and you do not need to use any other command.

Beam and Plate Bending

For beam bending, uniaxial stress prevails so that, $\sigma_x = E\,\varepsilon_x$. An alternate form of this equation that relates the bending moment with rotations is:

$$M = -EI\theta_x \quad \textbf{(6-7)}$$

For beam and truss elements, only the elastic modulus (EX) is used in the element stiffness matrix computation. Expanded in two dimensions, the above expression can be modified for plate bending as:

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \frac{Eh^3}{12(1-\nu)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{Bmatrix} \theta_x \\ \theta_y \\ \theta_{xy} \end{Bmatrix} \quad \textbf{(6-8)}$$

where h is the plate thickness. However, plate and shell elements available in COSMOS/M use the constitutive relations in the stress-strain form as opposed to the moment rotation form shown above.

Orthotropic Elastic Properties

In contrast to an isotropic material, an orthotropic material has preferred directions of strength which are mutually perpendicular. The properties along these directions (also known as principal directions) are the extreme values of elastic coefficients. The [C] or [D] matrix for an orthotropic material has nine independent elastic coefficients. In addition, there are three coefficients of thermal expansion.

In COSMOS/M, orthotropic material properties are applicable to all composite elements, SHELL3L, SHELL4L, SHELL9L, and SOLIDL, as well as the two dimensional continuum elements, PLANE2D and TRIANG.

Two-Dimensional Orthotropic Stress-Strain Relations

In two dimensions, the orthotropic stress-strain relations can be written as follows, including temperature effects:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{E_x}{1-\nu_{xy}\nu_{yx}} & \frac{\nu_{xy}E_y}{1-\nu_{xy}\nu_{yx}} & 0 \\ \frac{\nu_{xy}E_y}{1-\nu_{xy}\nu_{yx}} & \frac{E_y}{1-\nu_{xy}\nu_{yx}} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T-T_0) \begin{Bmatrix} \alpha_x \\ \alpha_y \\ 0 \end{Bmatrix} \right) \quad \textbf{(6-9)}$$

Note that in order to satisfy *symmetry* in the matrix of elastic moduli, $\nu_{xy}E_y = \nu_{yx}E_x$.

You need to therefore satisfy the foregoing symmetry condition when you input the numerical values of either the elastic modulus or Poisson's ratio.

Further, if you do *not* input the numerical value of the shear modulus, the program will compute it as shown below:

$$G_{xy}=\frac{E_x E_y}{E_x + E_y + 2 E_x \nu_{xy}} \qquad \textbf{(6-10)}$$

The procedure for defining an orthotropic material input is illustrated below. The illustration assumes that the program will internally calculate the shear modulus. To satisfy orthotropic symmetry relations, two elastic moduli and a Poisson's ratio are specified.

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **EX: Elasticity modulus in X mat. dir.**
Property value > **15.5E6**
Material Property Name > **EY: Elasticity modulus in Y mat. dir.**
Property value > **10.5E6**
Material Property Name > **NUXY: XY Poisson's ratio**
Property value > **0.25**
Material Property Name > **ALPX: Therm expansion coeff in X mat. dir.**
Property value > **0.25E-5**
Material Property Name > **ALPY: Therm expansion coeff in Y mat. dir.**
Property value > **0.45E-5**

The two dimensional orthotropic properties defined above will be used by the plane elements (PLANE2D and TRIANG) as long as all required elastic coefficients have been input. Otherwise, the program will default to using isotropic properties.

Three-Dimensional Orthotropic Stress-Strain Relations

In three dimensions, the orthotropic strain-stress relations take the form:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} 1/E_x & -\nu_{xy}/E_y & -\nu_{xz}/E_z & 0 & 0 & 0 \\ -\nu_{yx}/E_y & 1/E_y & -\nu_{yz}/E_y & 0 & 0 & 0 \\ -\nu_{zx}/E_z & -\nu_{zy}/E_y & 1/E_z & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{xy} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{yz} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{zx} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} \qquad \textbf{(6-11)}$$

In three dimensions, the orthotropic symmetry conditions dictate:

$$\frac{\nu_{xy}}{\nu_{yx}}=\frac{E_x}{E_y},\ \frac{\nu_{yz}}{\nu_{zy}}=\frac{E_y}{E_z},\ \frac{\nu_{zx}}{\nu_{xz}}=\frac{E_z}{E_x} \qquad \textbf{(6-12)}$$

When you input the orthotropic material properties in three dimensions, you must therefore make sure that the above symmetry conditions are *not* violated. The procedure for input of orthotropic elastic properties in three dimensions is similar to that demonstrated for two dimensions. Note that if you do not input the numerical values of shear moduli, the program will compute them using the relations shown below:

$$G_{xy}=\frac{E_x E_y}{E_x + E_y + 2 E_x \nu_{xy}},\ G_{yz} = G_{xz} = G_{xy} \qquad \textbf{(6-13)}$$

The default for ν_{xy} is 0.3 and for ν_{xz} and ν_{yz} is 0.0.

The three dimensional orthotropic properties defined above will be used by the elements SOLID and SOLIDL as long as all required elastic coefficients have been input. Otherwise, the program will default to using isotropic properties.

Axisymmetric Orthotropic Stress-Strain Relations

Axisymmetric solids with orthotropic material properties are also supported in COSMOS/M. Some times known as *cylindrically orthotropic*, the strain-stress relations for such material take the form:

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zr} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_r} & -\frac{\nu_{r\theta}}{E_\theta} & -\frac{\nu_{zr}}{E_z} & 0 \\ -\frac{\nu_{r\theta}}{E_r} & \frac{1}{E_\theta} & -\frac{\nu_{\theta z}}{E_z} & 0 \\ -\frac{\nu_{zr}}{E_r} & -\frac{\nu_{\theta z}}{E_\theta} & \frac{1}{E_z} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{zr}} \end{bmatrix} \begin{Bmatrix} \sigma_r \\ \sigma_\theta \\ \sigma_z \\ \tau_{zr} \end{Bmatrix} \qquad \textbf{(6-14)}$$

Note that equivalence for elastic coefficients in COSMOS/M between axisymmetric solids and general three dimensional solids is as follows: $E_r = E_x$, $E_\theta = E_z$, $E_z = E_y$. You can use this information for input of the orthotropic material data.

Notes on Input of Material Properties

General

- Ex must be given.
- If Ey and/or Ez are not given, they will be set equal to Ex.
- If Gxy is not given, it will be calculated from $G_{xy}=E_x \cdot E_y/(E_x+E_y+2NU_{xy} \cdot E_y)$.
- If Gyz and/or Gxz are not given, they will be set equal to Gxy.

PLANE 2D and TRIANG Elements

Isotropic

- Any given E's, G's, or NU's will be used even if not compatible.
- If any NU is not given, it will be assumed 0.3.
- If any G is not given, it will be set to zero.

Orthtropic

- If Ex, NUxy, and Gxy are given, they will be used even if not compatible.
- If Ex, Gxy, and NUxy are given, they will be used even if not compatible.
- If NUxy is not given, it will be assumed 0.3.
- If Gxy is not given, it will be calculated from $G_{xy}=E_x \cdot E_y/(E_x+E_y+2NU_{xy} \cdot E_y)$.

SOLID, TETRA4, TETRA4R, and TETRA10 Elements

Isotropic

- Given value of Gxy will be ignored. Gxy will always be computed from Gxy=Ex/[2(1+NUxy)] even if specified
- If NUxy is not given, it will be assumed 0.3.
- If you specify Ex and Gxy, Gxy will be re-calculated.

Orthotropic

- If Gxy, Gxz, and Gyz are given, they will be used even though not compatible.
- If Gxz, and Gyz are not given, they will be set equal to Gxy.
- If Gxy is not given, it will be calculated from $G_{xy}=E_x \cdot E_y/(E_x+E_y+2NU_{xy} \cdot E_y)$.

- The default for NUxy is 0.3
- The defaults for NUxz, and NUyz are 0.0.

SHELL3, SHELL4, SHELL3T, SHELL4T, SHELL9, and SHELLAX Elements

- User-defined Gxy is used only if the Shear Panel Option is supported and selected. For all other options, the given Gxy is ignored and will be calculated from Gxy=Ex/[2(1+NUxy)].
- The default value of NUxy is 0.3.

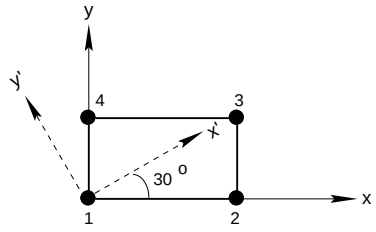
BEAM3D, PIPE, SOLIDL, and All Composite SHELLS,

- All entered values of G's are used even if not compatible.
- If not given, Gxy is set to zero.
- The default for NUxy is 0.3.

Orientation of Orthotropic Material Axes

By default, the orientation of the material coordinate axes for orthotropic materials is the *same* as that of the element local coordinate system (ECS =-1). You can specify an angle of rotation for the material coordinates *with respect to* the element local coordinate system while defining the section constants (**RCONST** (Propsets > **Real Constant**) command) for the applicable element. For example, if you are using 4-node plane element (PLANE2D) and would like to specify orthotropic material coordinate axes at an angle of 30 degrees with respect to the element local coordinates (see figure below), the procedure is illustrated below:

Figure 6-1. Specifying Orthotropic Material Axes in Element Local Coordinate System



Geo Panel: Propsets > **Real Constant**
Associated Element group [1] >
Real Constant set [1]>
Start location of the real constants [1] >
No. of real constants to be entered [2] >
RC1 : Thickness > **0.1**
RC2 : Material angle (Beta) [0] > **30**

In the figure above, the x-y axes represent the element local coordinates, and x'-y' represent the *rotated* orthotropic material coordinate axes. The above illustration is applicable to all elements with composite material description support except for SOLIDL for which the orthotropic material angle in a layer is specified under the RC4.

Anisotropic Elastic Properties

For general 3D anisotropic materials, there are no planes of symmetry and the elastic moduli are different in any direction. In three dimensions, the stress-strain relation for anisotropic materials involves 21 elastic constants as shown in the equation below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} & E_{14} & E_{15} & E_{16} \\ & E_{22} & E_{23} & E_{24} & E_{25} & E_{26} \\ & & E_{33} & E_{34} & E_{35} & E_{36} \\ & & & E_{44} & E_{45} & E_{46} \\ & \text{sym} & & & E_{55} & E_{56} \\ & & & & & E_{66} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \right) \quad \textbf{(6-15)}$$

Note that only the first three thermal expansion coefficients can be input in COSMOS/M and the remaining three are ignored. The property names of these coefficients considered are the same as those for orthotropic coefficients. The inverse form of the above relationship is written as follows:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} & E_{14} & E_{15} & E_{16} \\ & E_{22} & E_{23} & E_{24} & E_{25} & E_{26} \\ & & E_{33} & E_{34} & E_{35} & E_{36} \\ & & & E_{44} & E_{45} & E_{46} \\ & \text{sym} & & & E_{55} & E_{56} \\ & & & & & E_{66} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} \quad \textbf{(6-16)}$$

In COSMOS/M, the 3D anisotropic material property is currently supported for SOLID, SOLIDPZ, TETRA4, TETRA4R, TETRA10, and 2D anisotropic is available for SHELL3L, and SHELL4L elements only. You can input the coefficients of either the material stiffness or compliance matrix for anisotropic materials. All 21 material constants have unique names so that with suitable manipulations (explained a little later), you can describe materials with one or two planes of symmetry. If you need to model materials which are *monoclinic*, *transversely isotropic* or with *cubic* symmetry this feature becomes very useful.

For input purposes, the anisotropic material stiffness matrix [D] is referred to as the *standard* matrix, and material compliance matrix [C] is referred to as the *inverted* matrix. The illustration below demonstrates the procedure for entering the elastic *compliance* coefficients of an anisotropic material:

Geo Panel: Propsets > **Material Property (MPROP)**
Material property set [1] >
Material Property Name > **MC11: (1,1) stiffness matrix**
Property value [0] > **0.03E-6**
Material matrix type 1=Std 2=Inverted [1] > **2**
Material Property Name > **MC12: (1,2) stiffness matrix**
Property value [0] > **0.01E-6**
.
.
.
Material Property Name > **MC66: (1,3) stiffness matrix**
Property value [0] > **0.02E-6**
Material Property Name >

For 2D anisotropic, which is available for SHELL4L and SHELL3L elements only, equation (6-15) reduces to:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{14} & 0 & 0 \\ & E_{22} & E_{24} & 0 & 0 \\ & & E_{44} & 0 & 0 \\ & & & E_{55} & E_{56} \\ & \text{sym} & & & E_{66} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \right) \quad \textbf{(6-17)}$$

where the elastic coefficients are defined in the element material coordinates. Any consideration for the shear factor should be included in the specified values for E_{55}, E_{56}, and E_{66} by the user.

You need to specify the same material names as that for a fully anisotropic material and input only the non-zero coefficients.

In the case of a transversely isotropic material, the material exhibits a rotationally elastic symmetry about one of the coordinate axes. The number of independent constants in this case reduce to 5, and the stress-strain relations are as shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} & 0 & 0 & 0 \\ & E_{11} & E_{13} & 0 & 0 & 0 \\ & & E_{33} & 0 & 0 & 0 \\ & & & \frac{1}{2}(E_{11}-E_{12}) & 0 & 0 \\ & \text{sym} & & & E_{55} & 0 \\ & & & & & E_{55} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \quad \textbf{(6-18)}$$

In case of linearly elastic materials with cubic symmetry, the properties along the principal directions are identical. A material with cubic symmetry has only 3 independent elastic constants. The stress-strain equation for such a material is shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{12} & 0 & 0 & 0 \\ & E_{11} & E_{12} & 0 & 0 & 0 \\ & & E_{11} & 0 & 0 & 0 \\ & & & E_{44} & 0 & 0 \\ & \text{sym} & & & E_{44} & 0 \\ & & & & & E_{44} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \quad \textbf{(6-19)}$$

For a fully isotropic material, the form of the stress-strain equations is the same as that for a transversely isotropic material, but there are only two independent constants, E and ν. You can therefore use the format of input for anisotropic materials to describe isotropic materials by properly entering the elastic coefficients. However, this procedure is unwarranted and therefore not recommended for isotropic materials for which the input very simple as explained earlier under *Isotropic Elastic Properties*.

If you make use of the anisotropic material input format to describe isotropic materials, here is a word of caution. The program uses full integration in this case. The results will not be the same as those obtained using isotropic properties (defined using E and ν) which uses reduced integration by default.

Thermoelastic Properties

The effect of change in temperature on structures is two fold: variations in temperatures induce thermal strains affecting the state of stress; at the same time, temperature changes can also cause variations in the elastic material properties. Both effects can be efficiently treated in COSMOS/M. The following sections provide more information.

The table below summarizes the valid material properties in the Basic FEA System that can be input as temperature dependent.

Table 6-5. Temperature Dependent Material Properties in the Basic FEA System

Property Name	Description	Units
ALPX, ALPY, ALPZ	Coefficient of thermal expansion in X-, Y-, and Z-direction	Strain/Temp
DAMP	Material damping coefficient	
DCij	Dielectric material constants	
DENS	Mass density	Mass/Volume
EX, EY, EZ	Elastic modulus in X-, Y-, and Z-directions	Force/Area
GXY, GXZ, GYZ	Shear modulus, X-Y, X-Z, and Y-Z directions	*same as above*
MCij	Anisotropic elastic moduli	Force/Area
NUXY	Poisson's ratio, Y strain due to X strain	*none*
NUXZ	Poisson's ratio, Z strain due to X strain	*none*
NUYZ	Poisson's ratio, Z strain due to Y strain	*none*
PCij	Piezoelectric material coefficients	
SIGXC	Compressive strength in material X-direction	Force/Area
SIGXT	Tensile strength in material X-direction	*same as above*
SIGXYC	Compressive shear strength in the material X-Y plane	*same as above*
SIGXY	Shear strength in material X-Y plane	*same as above*
SIGYT	Tensile strength in material Y-direction	*same as above*

Thermal Stress-Strain Relations

Temperature variations inducing thermal stresses are treated as thermal loads in COSMOS/M. It is also possible to transfer temperature results from a heat transfer analysis as thermal loads in stress analysis for the same problem; see Chapter 7, *Loads and Boundary Conditions*, and Chapter 12, *Multidisciplinary Analyses*, for more information.

The relationship between thermal stresses and strains can be written in a *general* form as:

$$\{\sigma^t\} = [D]\{\varepsilon^t\} \quad \textbf{(6-20)}$$

where [D] is the elastic (isotropic, anisotropic, or orthotropic) material matrix, $\{\sigma^t\}$ is the thermal stress vector, and $\{\varepsilon^t\}$ is the thermal strain vector. The elastic moduli in [D] can be temperature dependent (discussed in the next section). Ignoring the effect of temperature on the coefficients of thermal expansion, the thermal strains are in general computed as:

$$\{\varepsilon^t\} = (T\text{-}T_0)\{\alpha^t\} \quad \textbf{(6-21)}$$

where $\{\alpha^t\}$ is the vector of thermal expansion coefficients, T is the existing (or imposed) temperature, and T_o is the reference temperature (defined by the **TREF** (LoadsBC > LOAD OPTIONS > **Reference Temp**) command) at which the structure is free of stress. Note that the reference temperature is also the temperature at which the coefficients of thermal expansion are usually specified. However, in COSMOS/M, the *coefficient of thermal expansion can be specified as temperature dependent*. The various equations presented earlier for isotropic, anisotropic, and orthotropic materials, also include the effect of thermal strains. The components of $\{\alpha^t\}$ for *thermally* isotropic, orthotropic, or anisotropic materials are:

$$\{\alpha^t\}_{\text{isotropic}} = \begin{Bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \\ 0 \\ 0 \end{Bmatrix}, \quad \{\alpha^t\}_{\text{orthotropic}} = \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_z \\ 0 \\ 0 \\ 0 \end{Bmatrix}, \quad \{\alpha^t\}_{\text{anisotropic}} = \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \quad \textbf{(6-22)}$$

For isotropic materials, you only need to specify ALPX for property name under the **MPROP** (Propsets > **Material Property**) command, and for orthotropic materials, you need to furnish the remaining two components, ALPY and ALPZ. However, in COSMOS/M, for anisotropic materials which require all six coefficients, *only* the first three values are allowed and these coefficients can be isotropic or orthotropic.

If the coefficients of thermal expansion are temperature dependent, you can use the **CURDEF** (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**) command to define a temperature versus thermal expansion coefficient curve, and associate

this curve with any component of $\{\alpha^t\}$. The following section illustrates in more detail.

Temperature Dependent Material Properties

Almost all elements in the Basic System can be used to model thermoelastic problems as indicated in Table 6-5. Many elastic material properties as well as the thermal expansion coefficients defined using the **MPROP** (Propsets > **Material Property**) command can be made temperature dependent. Typically, the elastic moduli *decrease* with increasing temperatures whereas Poisson's ratio remains constant. In case of stainless steel for example, the elastic modulus (E) decreases by about 20% when the temperature rises from 0° to 450° C.

To define the dependence of temperatures on elastic moduli, you can make use of the temperature curve definitions in COSMOS/M. First, you need to *activate* temperature curve definition using **ACTSET, TP, 1,** (Control > ACTIVATE > **Set Entity, Temperature Curve, 1**) and then use the **CURDEF** (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**) command to define different temperatures and the corresponding elastic moduli, as illustrated below. The stiffness matrix is formed using the interpolated material properties based on the temperature of the first considered load case in the analysis (i.e. with lowest load case label) regardless of the nature of that load case (i.e. whether it is a thermal load case or not). For this illustration, it was assumed that the elastic modulus decreases by 10% when the temperature increases from 0 to 100 degrees, and by 20% when it increases to 1000 degrees, i.e., a bilinear dependence of elastic modulus on temperatures. Note that you can either define a curve of temperature vs. multiplying factor for the elastic modulus, or, a curve of temperature vs. the elastic modulus.

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **Temp**
Curve number > **1**
Start point > **1**
Temperature for point 1 > **0**
Function value for point 1 > **1**
Temperature for point 2 > **100**
Function value for point 2 > **0.9**
Temperature for point 3 > **1000**
Function value for point 3 > **0.8**



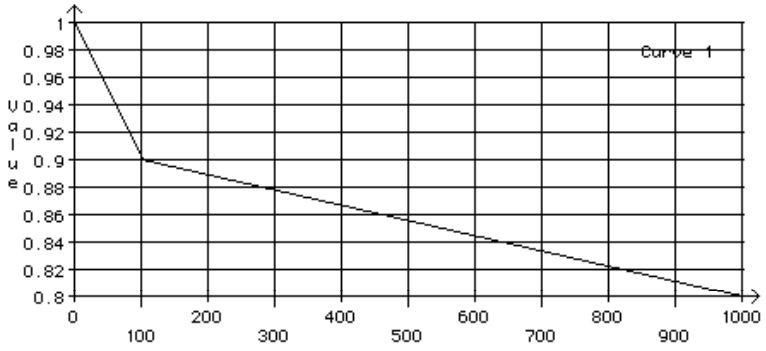

The above procedure associates the temperature curve labeled 1 with the dependencies defined. You can use the command **CURLIST** (LoadsBC > FUNCTION CURVE > **List Time/Temp**) to list the temperature dependencies on the elastic moduli defined above. The procedure for plotting a curve of temperature dependencies is illustrated below:

Geo Panel: Display > XY PLOTS > **Activate Pre-Proc**
Graph Number > **1**
Curve type > **Time/Temp/BHC/MPC**
Component > **Temp**
Curve number > **1**
Graph color > **12**
Graph line style > **Solid**
Graph symbol style > **Circle**

Geo Panel: Display > XY PLOTS > **Plot Curves**
Plot graph 1 0=No, 1=Yes > **Yes**

A plot as shown in the following figure will be displayed on the screen.

Figure 6-2. Example of Temperature Dependent Elastic Modulus



Next, to associate the temperature dependencies with the elastic modulus EX, use the command **MPROP** (Propsets > **Material Property**) and specify a multiplication factor for the property (the actual property values will be read from the temperature-elastic modulus curve defined earlier and multiplied by this factor) as illustrated below:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **EX: Elasticity modulus 1st dir**
Property value > **30E6**

If you list the material properties on the screen (using **MPLIST** (Edit > LIST > **Material Props**) command), the summary will be as shown below, confirming the associativity between EX and the temperature curve:

Label	Name	Temp/BH_Cr	Value
A 1	EX	1	3.000000e+07
A 1	EX	1	3.000000e+07
A 1	NUXY	0	3.300000e-01
A 1	GXY	0	5.800000e+06
A 1	ALPX	0	1.100000e-05
A 1	DENS	0	7.893000e-04
A 1	C	0	3.477600e+01
A 1	KX	0	5.370000e-03

If you continue defining other elastic moduli using the **MPROP** (Propsets > **Material Property**) command, they will also be associated with temperature-elastic modulus curve 1. To *deactivate* this dependency, you need to issue the command, **ACTSET, TP, 0,** (Control > ACTIVATE > **Set Entity, Temperature Curve, 0**).

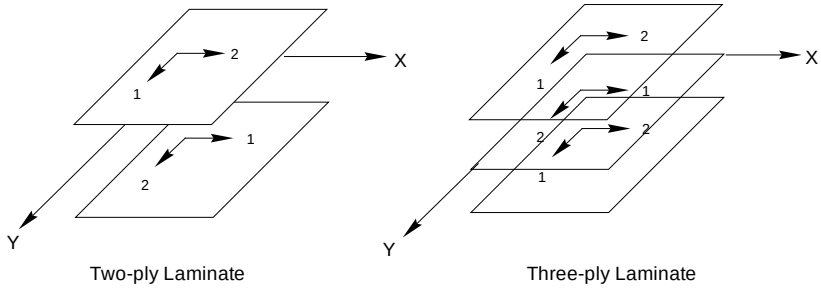
To define the dependence of temperatures on the coefficient of thermal expansion, you need to first *activate* temperature curve definition using **ACTSET, TP, 1,** (Control > ACTIVATE > **Set Entity, Temperature Curve, 1,)** and then execute the **CURDEF** (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**) command. This time, you can define the expansion coefficients vs. temperature values under temperature curve labeled 2 or any other non-zero integer, and follow this by the **MPROP** (Propsets > **Material Property**) command for defining ALPX or any other required component of thermal expansion coefficients.

To compute the effect of temperature on material properties, for each element the program uses one value of temperature which is the average of element nodal temperatures.

Laminated Composite Materials

Laminated composite materials consist of two or more *bonded* layers of different thicknesses and material properties. The material properties in each layer can be either isotropic or orthotropic. A common arrangement of the layers consists of two of the principal material axes of each layer oriented orthogonally with those of the next layer. Laminates with this configuration of alternating material directions are known as the *cross-ply* laminates. The figure below shows an exploded view of two- and three-ply laminates in cross-ply orientation.

Figure 6-3. Exploded View of Plane Laminated Structures with Cross-Ply Configuration



Two-ply Laminate Three-ply Laminate

A *stack* of laminae can be modeled with a single plate or shell element by the appropriate description of material properties of the stack in the element stiffness matrix. This is accomplished by utilizing the user-supplied information on number of layers, layer thickness, and relative orientation of orthotropic axes for each layer. The analysis results for each layer can be plotted and animated separately during postprocessing. The laminated composites can be modeled using three types of shell elements, SHELL3L, SHELL4L, SHELL9L, and the solid element, SOLIDL.

In COSMOS/M, the laminated composite is assumed to be made up of perfectly bonded laminae. The thickness of the bonds between laminae is assumed to be infinitesimally thin. The shear deformation at the bonds are negligible so that there is no interlaminar slip, and the displacements across the laminae boundaries are continuous.

✍ A laminated composite consisting of three laminae (with top and bottom layers of equal thickness and same material characteristics) is known as a sandwich plate or shell in COSMOS/M.

In order to use the sandwich option, you need to specify the appropriate flag under the **EGROUP** (Propsets > **Element Group**) command.

Material Properties for Laminated Composites

For laminated composites, the material properties of each layer is addressed by a material set number and the corresponding material constants are defined *as before* using the **MPROP** (Propsets > **Material Property**) command. A material set number may be referenced by one or more layers. The material properties in each layer may be orthotropic or isotropic. Please refer to the sections on isotropic and orthotropic material properties for more information on defining the elastic constants and the material axes. Note that the thickness of each layer, material group number and the orientation of material principal directions are specified under the **RCONST** (Propsets > **Real Constant**) command.

The procedure for defining the material properties of layered composites is as follows. It is recommended to first define the element group using the **EGROUP** (Propsets > **Element Group**) command and specify the composite element name and the number of layers in the element. The illustration below demonstrates this procedure for a laminated composite shell with four layers:

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **SHELL4L: Multilayer 4-node shell/plate element**

OP1:Type > **QUAD4**
OP2:Number of Layers > **4**
... ...
Accept

Next, define the required material properties using the **MPROP** (Propsets > **Material Property**) command. For a composite with four laminae, four different material groups are necessary if the properties for each layer are different. You can then proceed to defining the individual layer data using the **RCONST** (Propsets > **Real Constant**) command. In the illustration shown below, the material properties for all layers are the same, but the orientation of the material principal directions are different. For each layer, you will be prompted for the layer thickness, material group number, and material angle. The **RCONST** (Propsets > **Real Constant**) command accepts a maximum of 10 input data in each execution, and you therefore need to re-execute this command to input data for the remaining layers.

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**
Start location of the real constants > **1**
No. of real constants to be entered > **10**

RC1 : Dist. from ref. plane to top surface > **0.5**
RC2 : Temperature gradient > **0**
RC3 : Thickness of layer 1 > **0.25**
RC4 : Material number for layer 1 > **1**
RC5 : Material angle for layer 1 > **0**
RC6 : Thickness of layer 2 > **0.25**
RC7 : Material number for layer 2 > **1**
RC8 : Material angle for layer 2 > **30**
RC9 : Thickness of layer 3 > **0.25**
RC10 : Material number for layer 3 > **1**

Re-execute the **RCONST** (Propsets > **Real Constant**) command for the remaining data, as shown below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **11**
No. of real constants to be entered > **4**

RC11 : Material angle for layer 3 > **60**
RC12 : Thickness of layer 4 > **0.25**
RC13 : Material number for layer 4 > **1**
RC14 : Material angle for layer 4 > **90**

Failure Criteria for Laminated Composite Materials

A failure criterion provides the mathematical relation for strength under combined stresses. Unlike the conventional isotropic materials where one constant will suffice for failure stress level and location, laminated composite materials require more elaborate methods to establish failure stresses. The strength of a laminated composite can be based on the strength of individual plies within the laminate. In addition, the failure of plies can be successive as the applied load increases. There may be a first ply failure followed by other ply failures until the last ply fails, denoting the ultimate failure of the laminate. Progressive failure description is therefore quite complex for laminated composite structures.

A simpler approach for establishing failure consists of determining the structural integrity which depends on the definition of an allowable stress field. This stress field is usually characterized by a set of allowable stresses in the material *principal* directions. The table below lists the required components of allowable stresses referenced in the failure theories and their material names in COSMOS/M.

Table 6-6. Required Material Strength Components for Failure Criteria of Composites

Symbol	Description	Material Property Name
X_1^T	Tensile strength in the material longitudinal direction	SIGXT
X_1^C	Compressive strength in the material longitudinal direction	SIGXC
X_2^T	Tensile strength in the material transverse direction	SIGYT
X_2^C	Compressive strength in the material transverse direction	SIGYC
S_{12}	In-plane shear strength in the material x-y plane	SIGXY

Two failure criteria which make use of the above allowable stresses input by the user are currently available. These include the Hill, and the Tsai-Wu failure criteria. These failure criteria are used to calculate a failure index (F.I.) from the computed stresses and user-supplied material strengths. A failure index of 1 denotes the onset of failure, and a value less than 1 denotes no failure. The following sections provide more information on the two failure criteria available.

The failure criterion of Hill or Tsai-Wu is applicable to SHELL3L and SHELL4L elements *only*. All components of material strengths for all layers ***must*** be input in order to compute the failure indices.

The failure indices are computed for all layers in each element of your model. During postprocessing, you can plot failure indices of your mesh for any layer.

Hill Failure Criterion

The Hill failure criterion (sometimes known as the Tsai-Hill criterion) is frequently applied to orthotropic materials with *equal* strengths in tension and compression. The failure index computed according the Hill theory shown below represents the analytical expression for an ellipse in the σ_1 - σ_2 plane:

$$\text{F.I.} = \frac{\sigma_1^2}{X_1^2} - \frac{\sigma_1\sigma_2}{X_1^2} + \frac{\sigma_2^2}{X_2^2} + \frac{\tau_{12}^2}{S_{12}^2} \qquad \textbf{(6-23)}$$

where σ_1, σ_2 and τ_{12} are the computed stresses in the material directions, (SIGMA-X, SIGMA-Y and SIGMA-XY in the output file) and,

$$X_1 = X_1^T \text{ if } \sigma_1 > 0; \quad X_1 = X_1^C \text{ if } \sigma_1 < 0$$

$$X_2 = X_2^T \text{ if } \sigma_2 > 0; \quad X_2 = X_2^C \text{ if } \sigma_2 < 0$$

The application of Hill failure criterion is requested by specifying a value of 2 under the **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) command as illustrated below:

Control Panel: Analysis > STATIC > **Stress Analysis Options**
AISC code check Flag > **No**
... ...
Type of failure analysis > **Hill Criterion**

The program computes the failure indices for all layers of each element and prints them along with the element stresses in the output file as shown below. For the listing shown below, SHELL3L elements were used with four layers.

```
 ELEMENT NUMBER    1

      NX         NY         NXY         MX         MY         MXY        VX         VY
 -2.70E+03 -1.04E+03 -4.39E+01 -3.75E+03 -2.73E+02 -6.51E+02 -1.26E+02 -1.21E+02


  LAYER  (ECS:-1) SIGMA-X     SIGMA-Y    SIGMA-XY     TAU-XZ       TAU-YZ          HILL
NUMBER
   4     TOP     -2.941E+04  -4.544E+03  -4.193E+03  -1.322E+02  -1.551E+02    4.93245E+01
         BOTTOM  -1.686E+04  -3.564E+03  -1.452E+03  -1.322E+02  -1.551E+02    1.34369E+01
                 (INTERLAMINAR SHEAR STRESSES)       -1.395E+02  -1.350E+02
   3     TOP     -2.621E+04   3.514E+03  -6.611E+03  -1.526E+02  -1.611E+02    1.23329E+02
         BOTTOM  -5.560E+03  -1.424E+03  -4.318E+02  -1.526E+02  -1.611E+02    2.45260E+00
                 (INTERLAMINAR SHEAR STRESSES)       -2.231E+02  -1.774E+02
   2     TOP     -7.103E+03  -5.668E+02   9.327E+02  -1.679E+02  -1.464E+02    6.25082E+00
         BOTTOM   1.537E+04  -4.018E+03   5.256E+02  -1.679E+02  -1.464E+02    4.80744E+01
                 (INTERLAMINAR SHEAR STRESSES)       -1.697E+02  -1.109E+02
   1     TOP      1.240E+04  -5.105E+02   4.030E+03  -1.630E+02  -1.258E+02    3.28336E+01
         BOTTOM   3.103E+04   9.984E+02   6.771E+03  -1.630E+02  -1.258E+02    1.69770E+0

      ENRGYE = 0.5509E+03  ENRGYD = 0.1344E+02
```

Tsai-Wu Failure Criterion

The Tsai-Wu failure criterion (also known as the Tsai-Wu tensor polynomial theory) is commonly used for orthotropic materials with *unequal* tensile and compressive strengths. The failure index according to this theory is computed using the following equation:

$$\text{F.I.} = \left(F_1\sigma_1 + F_2\sigma_2\right) + \left(F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + 2F_{12}\sigma_1\sigma_2 + F_{66}\tau_{12}^2\right)$$

where,

$$F_1 = \frac{1}{X_1^T} - \frac{1}{X_1^C}, \qquad F_2 = \frac{1}{X_2^T} - \frac{1}{X_2^C}, \qquad F_{11} = \frac{1}{X_1^T X_1^C},$$

$$F_{22} = \frac{1}{X_2^T X_2^C}, \qquad F_{66} = \frac{1}{S_{12}^2} \qquad \textbf{(6-24)}$$

and

$$F_{12} = -\frac{1}{2}\sqrt{\frac{1}{X_1^T X_1^C X_2^T X_2^C}} \qquad \textbf{(6-25)}$$

The application of Tsai-Wu failure criterion is requested by specifying a value of 1 under the **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) command as illustrated below:

Geo Panel: Analysis > STATIC > **Stress Analysis Options**
AISC code check Flag > **No**
... ...
Type of failure analysis > **Tsai-Wu Criterion**

The program computes the failure indices for all layers in each element and prints them along with the element stresses in the output file as shown below. The listing shown below is for the same example as that for the Hill criterion using SHELL3L elements with four layers. You can notice the change in value of the failure indices.

```
 ELEMENT NUMBER      1

      NX        NY        NXY         MX         MY        MXY         VX         VY
 -2.70E+03 -1.04E+03 -4.39E+01 -3.75E+03 -2.73E+02 -6.51E+02 -1.26E+02 -1.21E+02


  LAYER (ECS:-1)    SIGMA-X     SIGMA-Y     SIGMA-XY      TAU-XZ       TAU-YZ       TSAI-WU
NUMBER
    4     TOP      -2.941E+04  -4.544E+03  -4.193E+03  -1.322E+02  -1.551E+02   1.11044E+02
       BOTTOM      -1.686E+04  -3.564E+03  -1.452E+03  -1.322E+02  -1.551E+02   3.37088E+01
                  (INTERLAMINAR SHEAR STRESSES)        -1.395E+02  -1.350E+02
    3     TOP      -2.621E+04   3.514E+03  -6.611E+03  -1.526E+02  -1.611E+02   1.46136E+02
       BOTTOM      -5.560E+03  -1.424E+03  -4.318E+02  -1.526E+02  -1.611E+02   3.78808E+00
                  (INTERLAMINAR SHEAR STRESSES)        -2.231E+02  -1.774E+02
    2     TOP      -7.103E+03  -5.668E+02   9.327E+02  -1.679E+02  -1.464E+02   7.79441E+00
       BOTTOM       1.537E+04  -4.018E+03   5.256E+02  -1.679E+02  -1.464E+02   4.63253E+01
                  (INTERLAMINAR SHEAR STRESSES)        -1.697E+02  -1.109E+02
    1     TOP       1.240E+04  -5.105E+02   4.030E+03  -1.630E+02  -1.258E+02   2.84385E+01
       BOTTOM       3.103E+04   9.984E+02   6.771E+03  -1.630E+02  -1.258E+02   1.51238E+02

  ENRGYE = 0.5509E+03  ENRGYD = 0.1344E+02
```

Piezoelectric Properties

The term piezoelectricity refers to the phenomenon of electrical polarization of some crystals due to deformation in certain directions. The reverse phenomenon, namely the mechanical deformations of crystals due to an applied electric field, is also termed as piezoelectricity. It is the latter phenomenon that is widely used in study of piezoelectric resonators and many other applications. A piezoelectric resonator for example, is a device consisting of a bar or a plate made from a piezoelectric material with two or more electrodes attached to its surface. When a harmonic electric voltage is applied to the electrodes, the piezoelectric material undergoes mechanical vibrations. For resonance, the exciting signal frequency (of the electric voltage) must be close to the resonant frequency. Piezoelectric effect also plays an important role in the fields of crystal physics and transducer technology. The type of finite element analyses you can perform on a piezoelectric material include static, modal (natural frequency), harmonic, and transient analysis. The term *electroelastic* analysis is frequently used for the study of deformations in piezoelectric materials subjected to electric fields.

COSMOS/M provides you the capability to analyze the free vibrations of piezoelectric components subjected to an electric field. A new solid element (SOLIDPZ) with isotropic, anisotropic or orthotropic elastic material properties can be used in the electroelastic analysis of piezoelectric components.

Piezoelectric Materials

A common example of piezoelectric material is the Quartz crystal used in wristwatches and other time-keeping devices. However, the application of piezoelectric materials has grown to numerous other areas. Piezoelectric materials today are not restricted to crystalline forms alone, and they extend to ceramics and polymers. Modern piezoelectric ceramics is based on solid solutions of lead zirconate ($PbZrO_3$) and lead titanate ($PbTiO_3$) and is referred to as PZT ceramics. The table below provides examples of some commonly used piezoelectric materials in the industry.

Table 6-7. Examples of Piezoelectric Materials

Type of Piezoelectric Material	Examples	Remarks
Piezoelectric Crystalline Materials	Quartz Berlinite Lithium Niobate Lithium Tantalate Bismuth Germanium Oxide Bismuth Silicium Oxide	. . . .
Piezoelectric Ceramics	PZT VIBRIT PKM	Clevite Corporation Siemens . . .
Piezoelectric Polymers	Polyvinyl Chloride (PVC) Polyvinyl Fluoride (PVF) Difluor Polyethylene (PVF_2) Nylon 11	. . . . . . PVF_2-β and PVF_2-δ are the two common forms. . . .

Piezoelectric Property Definitions in COSMOS/M

In order to perform electroelastic analysis, the *elastic*, *dielectric*, and *piezoelectric* properties of the piezoelectric material need to input. Many piezoelectric materials have crystalline structure with well defined planes of strength and symmetry which often render the isotropic elastic description inadequate. Orthotropic or anisotropic elastic properties need to be used. However, isotropic elastic

property definitions are also supported for the SOLIDPZ element in addition to the orthotropic and anisotropic properties. The elastic properties can be input as explained before.

In electroelastic analysis, the elastic constitutive equations (equation 6-1) and the electric field constitutive equations *(see Advanced Modules Manual)* are coupled by the piezoelectric matrix. Therefore, the electromechanical constitutive equations for linear behavior are written as follows:

$$\{\sigma\} = [D]\{\varepsilon\} - [e]\{E_1\}$$
$$\{D_1\} = [e]^T\{\varepsilon\} + [\varepsilon_d]\{E_1\} \quad \textbf{(6-26)}$$

where $\{\sigma\}$ is the stress vector, $\{D_1\}$ is a vector of electric flux density, [D] as before is the matrix of elastic coefficients at *constant* electric field (free of electrostrictive effects), $\{\varepsilon\}$ represents the mechanical strain, [e] is a matrix of piezoelectric properties, $\{E_L\}$ is the vector representing the electric field, and $[\varepsilon_d]$ is the dielectric property matrix at *constant* mechanical strain. The coefficients of the elastic material matrix [D] are input as explained earlier, using isotropic, orthotropic, or anisotropic material descriptions. The following paragraphs describe the input for piezoelectric and dielectric properties.

The piezoelectric matrix [e] relates the electric field to stress, and it is a 6x3 matrix for 3D, and 4x2 matrix for 2D models. This matrix is shown below for 3D models.

$$[e] = \begin{bmatrix} e_{11} & e_{12} & e_{13} \\ e_{21} & e_{22} & e_{23} \\ e_{31} & e_{32} & e_{33} \\ e_{41} & e_{42} & e_{43} \\ e_{51} & e_{52} & e_{53} \\ e_{61} & e_{62} & e_{63} \end{bmatrix} \quad \textbf{(6-27)}$$

As seen in Table 6-2, the property names of the coefficients in the piezoelectric matrix above are denoted as PCij where i varies from 1 to 6 and j varies from 1 to 3. Therefore, to define piezoelectric material properties, the **MPROP** (Propsets > **Material Property**) command is used as follows:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **PC11: (1,1) Piezoelectric matrix**
Property value [0]> **-4.1**
... ...

Material Property Name > **PC63: (1,3) Piezoelectric matrix**
Property value [0] > **14.1**

The dielectric property matrix $[\varepsilon_d]$ is shown below:

$$[\varepsilon_d]=\begin{bmatrix} d_{11} & d_{12} & d_{13} \\ d_{21} & d_{22} & d_{13} \\ d_{31} & d_{32} & d_{33} \end{bmatrix} \quad \textbf{(6-28)}$$

The property names of the coefficients in the above matrix are denoted as DCij where i and j vary from 1 to 3. To define dielectric material properties, the **MPROP** (Propsets > **Material Property**) command is used as follows:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **DC11: (1,1) Dielectric matrix**
Property value > **7.124E-9**
... ...
Material Property Name > **DC33: (1,1) Dielectric matrix**
Property value > **5.85E-9**

Mass and Damping Properties

The description of mass properties requires the input of mass density which is specified by the property name DENS under the **MPROP** (Propsets > **Material Property**) command. This data is used for computing the element lumped or distributed mass matrix used in calculating natural frequencies. Mass data is also required if you want to apply gravity and centrifugal loading.

Concentrated mass at a node can be defined using a mass element; see Chapter 5, *Element Library* for more information.

Damping properties are specified using the property name DAMP under the **MPROP** (Propsets > **Material Property**) command. The inclusion of damping information results in the extraction of complex eigenvalues. Note that damping can be applied to models with 50 or fewer degrees of freedom in the Basic FEA System.

Inelastic Material Behavior

In addition to the various types of elastic, thermoelastic, composite, and piezo-electric material properties discussed so far, COSMOS/M features extensive inelastic, time and temperature dependent material descriptions for nonlinear structural, thermal, and electromagnetic analyses. The inelastic material properties can ***only*** be used in nonlinear analysis using the NSTAR program. For more information, please refer to the COSMOS/M Advanced Modules Manual - Part 1, *NSTAR Chapter 3*.

Material Library

The material library feature of COSMOS/M allows you to either select standard material properties from a built-in library, or, your own material properties from a user-defined material library. The built-in material properties are available in FPS, MKS, or SI system of units. The property names used in the material libraries and their description is provided in the table below.

Table 6-8. Property Names in the Material Libraries

Property Name	Symbol	Description
ALPX	α_x	Coefficient of thermal expansion in the x-direction
BETA	β	Coefficient of volume expansion
C (Cp)	C_p	Specific heat
DENS	ρ	Mass density
EX	E_x	Elastic modulus in the x-direction
GAMMA	γ	Ratio of specific heats
GXY	G_{xy}	Shear modulus in the x-y plane
KX	K_x	Thermal conductivity in the x-direction
NUXY	ν_{xy}	Poisson's ratio for x-strain due to y-stress
VISC	η	Dynamic viscosity

By default, both built-in and user-defined material libraries assume the material properties to be *isotropic*. If it is necessary to use orthotropic material properties in your model, you need to therefore provide the remaining property components under the same material group. However, it is not possible to append the material library properties for describing anisotropic materials at this time.

The commands **PICK_MAT** (Propsets > **Pick Material Lib**) and **USER_MAT** (Propsets > **User Material Lib**) can be used for selecting the pre-defined material properties from the built-in or user-defined material libraries respectively. The following sections describe the procedure for using the material libraries in more detail.

Using the Built-In Material Library

For some of the commonly used types of materials, the properties have been pre-defined in COSMOS/M. For materials with isotropic, linear elastic and linear heat transfer properties, you can use the built-in material library using the **PICK_MAT** (Propsets > **Pick Material Lib**) command.

When you use the **PICK_MAT** (Propsets > **Pick Material Lib**) command, the material properties are read from a text file, PICKMAT.LIB, installed in the COSMOS/M directory. The types of materials available and the corresponding names in COSMOS/M are listed in the table below:

Table 6-9. Available Material Types in the Built-In Library

COSMOS/M Name	Material Type
ACRYLIC	Medium-High Impact Acrylic
AIR	Air (300 K)
ALUMINUM	Aluminum Alloy
AL_1345	Aluminum 1345 Alloy
AL_1350	Aluminum 1350 Alloy
AL_2014	Aluminum 2014 Alloy
AL_2018	Aluminum 2018 Alloy
AL_2024	Aluminum 2024 Alloy
AL_3003	Aluminum 3003 Alloy
AL_6061	Aluminum 6061 Alloy
AL_7079	Aluminum 7079 Alloy
AL_BRONZE	Aluminum Bronze
A_STEEL	Alloy Steel
BRASS	Silicon Brass & Bronze
CA_STEEL	Cast Alloy Steel, below 8% alloy content
COBALT	Cobalt

Table 6-9. Available Material Types in the Built-In Library (Concluded)

COSMOS/M Name	Material Type
CS_STEEL	Cast Stainless Steel, CF-8M or CF-20, water quenched
D_NICKEL	Nickel Alloy, Duranickel 301
GC_IRON	Gray Cast Iron, ASTM Class 40
GLASS	Glass
GOLD	Pure Gold
IRON	Iron
LEAD	Pure Lead
MAGNES	Magnesium Alloy, wrought or cast
MC_IRON	Malleable Cast Iron, ASTM A220
MN_BRONZE	Manganese Bronze
MOLYBDENUM	Molybdenum
MONEL	MONEL 400 alloy of Nickel
NYLON	Nylon 6/10
PC_STEEL	Plain Carbon Steel
PORCELAIN	Ceramic Porcelain
RUBBER	Rubber
SILVER	Pure Silver
STEEL	Steel
ST_1020	Steel, AISI C1020 (Hot Worked)
ST_304	Steel, AISI 304 (Sheet)
ST_ST	Stainless Steel
TITANIUM	Titanium
TUNGSTEN	Tungsten
T_BRONZE	Tin Bronze 1B
VANADIUM	Vanadium
WATER	Water (300 K)
WS_STEEL	Wrought Stainless Steel
W_COPPER	Wrought Copper
ZIRCONIUM	Zirconium

The properties in the built-in material library are valid at a temperature of 70° F. Some of the properties can be made temperature dependent; see the section on thermoelastic properties.

Using the built-in material library is *extremely* simple. For example, if you like to select Wrought Copper using the FPS system of units for material set 1, the **PICK_MAT** (Propsets > **Pick Material Lib**) command is executed as follows:

Geo Panel: Propsets > **Pick Material Lib**
Material property set > **1**
Material Name > **W_COPPER: Wrought Copper**
Unit Label > **FPS: British (Inch/Pound/Second)**

The properties of Wrought Copper stored in the library are shown below:

Material Name: W_COPPER Wrought Cooper			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.16E+08 psi	0.11E+07 Kgf/cm/cm	0.11E+12 Pascals
NUXY	0.33	0.33	0.33
GXY	0.58E+07 psi	0.41E+06 Kgf/cm/cm	0.40E+11 Pascals
ALPX	0.11E-04 /Fahrenheit	0.20E-04 /Centigrade	0.20E-04 /Kelvin
DENS	0.79E-03 $lbf\,s^2/in^4$	0.86E-05 $Kgf\,s^2/cm^4$	0.84E+04 Kgm/m^3
KX	0.31E-02 BTU/in/s/F	0.56 Cal/cm/s/C	0.23E+03 W/m/K
C (Cp)	35.0 BTU in/lbf/s/s/F	0.88E+05 Cal cm/kgf/s/s/C	0.38E+03 J/kgm/K

For linear static stress analysis, the isotropic stress-strain relations (including temperature effects) for the selected material in FPS units under plane stress conditions are as follows:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = 17.3942 \times 10^6 \begin{bmatrix} 1 & 0.33 & 0 \\ 0.33 & 1 & 0 \\ 0 & 0 & 0.335 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - 11.0 \times 10^{-6} (T - T_0) \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \right)$$

Appending the above material properties to describe orthotropy is straight forward. For example, if you know the remaining orthotropic properties of copper (we are *not* suggesting here that copper is a orthotropic material!) and would like to use them, the procedure for appending the library properties for the remaining property components is illustrated below:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **EY: Elasticity modulus in Y mat. dir.**
Property value > **20.5E6**
Material Property Name > **ALPY: Therm expansion coeff in Y mat. dir.**
Property value > **15.5E-6**
Material Property Name > **KY: Thermal conductivity in Y mat. dir.**
Property value > **8.5E-3**

Note that it is necessary to input either NUYX (Poisson's ratio for y-strain due to x-stress) or EY (elastic modulus in the y-direction) as the program will internally compute one of these quantities using the symmetry condition, $\nu_{xy} E_y = \nu_{yx} E_x$. With the properties defined above, the orthotropic stress-strain relations for Copper take the following form:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = 10^6 \begin{bmatrix} 18.13 & 7.91 & 0 \\ 7.91 & 23.98 & 0 \\ 0 & 0 & 5.8 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - 10^{-6} (T - T_0) \begin{Bmatrix} 11 \\ 15.5 \\ 0 \end{Bmatrix} \right)$$

The above equations are used in the program for computing the element stiffness matrices and in other calculations.

You can use the command **MPLIST** (Edit > LIST > **Material Props**) to list the properties defined (as well as those selected from the library) on the screen for verification, as shown below:

```
Label      Name    Temp/BH_Cr          Value
A    1       EX             0      1.550000e+07
A    1       EY             0      2.050000e+07
A    1     NUXY             0      3.300000e-01
A    1      GXY             0      5.800000e+06
A    1     ALPX             0      1.100000e-05
A    1     ALPY             0      1.550000e-05
A    1     DENS             0      7.893000e-04
A    1        C             0      3.477600e+01
A    1       KX             0      5.370000e-03
A    1       KY             0      8.500000e-03
```

Contents of the Built-In Material Library

The material properties have been pre-defined for 45 materials in the built-in library. The properties in all three system of units are listed in the following paragraphs.

The properties in the built-in material library are valid at a temperature of 70° F. You need to make corrections if necessary for any other set of reference conditions, especially for temperature sensitive properties such as the coefficient of thermal expansion.

Material Name: A_STEEL Alloy Steel					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.30E+08	psi	0.21E+07	Kgf/cm/cm	0.21E+12 Pascals
NUXY	0.28		0.28		0.28
GXY	0.12E+08	psi	0.81E+06	Kgf/cm/cm	0.79E+11 Pascals
ALPX	0.74E-05	/Fahrenheit	0.13E-04	/Centigrade	0.13E-04 /Kelvin
DENS	0.73E-03	$lbf\,s^2/in^4$	0.79E-05	$Kgf\,s^2/cm^4$	0.77E+04 Kgm/m^3
KX	0.67E-03	BTU/in/s/F	0.12	Cal/cm/s/C	0.46E+03 J/kgm/K
C (Cp)	42.0	BTU in/lbf/s/s/F	0.11E+06	Calscm/kgf/s/s/C	0.46E+03 J/kgm/K
Material Name: ACRYLIC Acrylic (Medium-High Impact)					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.35E+06	psi	0.25E+05	Kgf/cm/cm	0.24E+10 Pascals
NUXY	0.35		0.35		0.35
GXY	0.13E+06	psi	0.91E+04	Kgf/cm/cm	0.89E+09 Pascals
ALPX	0.29E-04	/Fahrenheit	0.52E-04	/Centigrade	0.52E-04 /Kelvin
DENS	0.11E-03	$lbf\,s^2/in^4$	0.12E-05	$Kgf\,s^2/cm^4$	0.12E+04 Kgm/m^3
KX	0.28E-05	BTU/in/s/F	0.50E-03	Cal/cm/s/C	0.21 W/m/K
C (Cp)	0.14E+03	BTU in/lbf/s/s/F	0.34E+06	Cal cm/kgf/s/s/C	0.15E+04 J/kgm/K
Material Name: ALUMINUM Aluminum Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11 Pascals
ALPX	0.13E-04	/Fahrenheit	0.24E-04	/Centigrade	0.24E-04 /Kelvin
DENS	0.25E-03	$lbf\,s^2/in^4$	0.28E-05	$Kgf\,s^2/cm^4$	0.27E+04 Kgm/m^3
KX	0.27E-02	BTU/in/s/F	0.49	Cal/cm/s/C	0.20E+03 W/m/K
C (Cp)	83.0	BTU in/lbf/s/s/F	0.21E+06	Cal cm/kgf/s/s/C	0.90E+03 J/kgm/K

Material Name: AL_1345 Aluminum 1345 Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11 Pascals
ALPX	0.13E-04	/Fahrenheit	0.24E-04	/Centigrade	0.24E-04 /Kelvin
DENS	0.25E-03	$lbf\,s^2/in^4$	0.27E-05	$Kgf\,s^2/cm^4$	0.27E+04 Kgm/m^3
KX	0.29E-02	BTU/in/s/F	0.53	Cal/cm/s/C	0.22E+03 W/m/K
C (Cp)	93.0	BTU in/lbf/s/s/F	0.24E+06	Cal cm/kgf/s/s/C	0.10E+04 J/kgm/K
Material Name: AL_1350 Aluminum 1350 Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11 Pascals
ALPX	0.13E-04	/Fahrenheit	0.24E-04	/Centigrade	0.24E-04 /Kelvin
DENS	0.25E-03	$lbf\,s^2/in^4$	0.28E-05	$Kgf\,s^2/cm^4$	0.27E+04 Kgm/m^3
KX	0.31E-02	BTU/in/s/F	0.56	Cal/cm/s/C	0.23E+03 W/m/K
C (Cp)	93.0	BTU in/lbf/s/s/F	0.24E+06	Cal cm/kgf/s/s/C	0.10E+04 J/kgm/K
Material Name: AL_2014 Aluminum 2014 Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.11E+08	psi	0.78E+06	Kgf/cm/cm	0.77E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.40E+07	psi	0.28E+06	Kgf/cm/cm	0.28E+11 Pascals
ALPX	0.13E-04	/Fahrenheit	0.23E-04	/Centigrade	0.23E-04 /Kelvin
DENS	0.26E-03	$lbf\,s^2/in^4$	0.28E-05	$Kgf\,s^2/cm^4$	0.28E+04 Kgm/m^3
KX	0.21E-02	BTU/in/s/F	0.38	Cal/cm/s/C	0.16E+03 W/m/K
C (Cp)	89.0	BTU in/lbf/s/s/F	0.23E+06	Cal cm/kgf/s/s/C	0.96E+03 J/kgm/K
Material Name: AL_2018 Aluminum 2018 Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.11E+08	psi	0.76E+06	Kgf/cm/cm	0.74E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11 Pascals
ALPX	0.12E-04	/Fahrenheit	0.22E-04	/Centigrade	0.22E-04 /Kelvin
DENS	0.26E-03	$lbf\,s^2/in^4$	0.29E-05	$Kgf\,s^2/cm^4$	0.28E+04 Kgm/m^3
KX	0.21E-02	BTU/in/s/F	0.37	Cal/cm/s/C	0.15E+03 W/m/K
C (Cp)	93.0	BTU in/lbf/s/s/F	0.24E+06	Cal cm/kgf/s/s/C	0.10E+04 J/kgm/K
Material Name: AL_2024 Aluminum 2024 Alloy					
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**
EX	0.11E+08	psi	0.74E+06	Kgf/cm/cm	0.73E+11 Pascals
NUXY	0.33		0.33		0.33
GXY	0.40E+07	psi	0.28E+06	Kgf/cm/cm	0.28E+11 Pascals
ALPX	0.13E-04	/Fahrenheit	0.23E-04	/Centigrade	0.23E-04 /Kelvin
DENS	0.26E-03	$lbf\,s^2/in^4$	0.28E-05	$Kgf\,s^2/cm^4$	0.28E+04 Kgm/m^3
KX	0.19E-02	BTU/in/s/F	0.34	Cal/cm/s/C	0.14E+03 W/m/K
C (Cp)	73.0	BTU in/lbf/s/s/F	0.19E+06	Cal cm/kgf/s/s/C	0.80E+03 J/kgm/K

Material Name: AL_3003 Aluminum 3003 Alloy							
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11	Pascals
NUXY	0.33		0.33		0.33	
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11	Pascals
ALPX	0.13E-04	/Fahrenheit	0.23E-04	/Centigrade	0.23E-04	/Kelvin
DENS	0.26E-03	lbf s^2/in^4	0.28E-05	Kgf s^2/cm^4	0.27E+04	Kgm/m^3
KX	0.22E-02	BTU/in/s/F	0.40	Cal/cm/s/C	0.17E+03	W/m/K
C (Cp)	93.0	BTU in/lbf/s/s/F	0.24E+06	Cal cm/kgf/s/s/C	0.10E+04	J/kgm/K
Material Name: AL_6061 Aluminum 6061 Alloy						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11	Pascals
NUXY	0.33		0.33		0.33	
GXY	0.38E+07	psi	0.27E+06	Kgf/cm/cm	0.26E+11	Pascals
ALPX	0.13E-04	/Fahrenheit	0.24E-04	/Centigrade	0.24E-04	/Kelvin
DENS	0.25E-03	lbf s^2/in^4	0.28E-05	Kgf s^2/cm^4	0.27E+04	Kgm/m^3
KX	0.22E-02	BTU/in/s/F	0.40	Cal/cm/s/C	0.17E+03	W/m/K
C (Cp)	0.12E+03	BTU in/lbf/s/s/F	0.29E+06	Cal cm/kgf/s/s/C	0.13E+04	J/kgm/K
Material Name: AL_7079 Aluminum 7079 Alloy						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.10E+08	psi	0.73E+06	Kgf/cm/cm	0.72E+11	Pascals
NUXY	0.33		0.33		0.33	
GXY	0.39E+07	psi	0.27E+06	Kgf/cm/cm	0.27E+11	Pascals
ALPX	0.14E-04	/Fahrenheit	0.25E-04	/Centigrade	0.25E-04	/Kelvin
DENS	0.26E-03	lbf s^2/in^4	0.28E-05	Kgf s^2/cm^4	0.27E+04	Kgm/m^3
KX	0.16E-02	BTU/in/s/F	0.29	Cal/cm/s/C	0.12E+03	W/m/K
C (Cp)	89.0	BTU in/lbf/s/s/F	0.23E+06	Cal cm/kgf/s/s/C	0.96E+03	J/kgm/K
Material Name: AL_BRONZE Aluminum Bronze						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.17E+08	psi	0.12E+07	Kgf/cm/cm	0.11E+12	Pascals
NUXY	0.30		0.30		0.30	
GXY	0.62E+07	psi	0.44E+06	Kgf/cm/cm	0.43E+11	Pascals
ALPX	0.95E-05	/Fahrenheit	0.17E-04	/Centigrade	0.17E-04	/Kelvin
DENS	0.70E-03	lbf s^2/in^4	0.76E-05	Kgf s^2/cm^4	0.74E+04	Kgm/m^3
KX	0.75E-03	BTU/in/s/F	0.13	Cal/cm/s/C	56.	W/m/K
C (Cp)	35.0	BTU in/lbf/s/s/F	0.88E+05	Cal cm/kgf/s/s/C	0.38E+03	J/kgm/K
Material Name: BRASS Silicon Brass and Bronze						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.15E+08	psi	0.11E+07	Kgf/cm/cm	0.10E+12	Pascals
NUXY	0.33		0.33		0.33	
GXY	0.54E+07	psi	0.38E+06	Kgf/cm/cm	0.37E+11	Pascals
ALPX	0.10E-04	/Fahrenheit	0.18E-04	/Centigrade	0.18E-04	/Kelvin
DENS	0.80E-03	lbf s^2/in^4	0.87E-05	Kgf s^2/cm^4	0.85E+04	Kgm/m^3
KX	0.15E-02	BTU/in/s/F	0.26	Cal/cm/s/C	0.11E+03	W/m/K
C (Cp)	36.0	BTU in/lbf/s/s/F	0.90E+05	Cal cm/kgf/s/s/C	0.39E+03	J/kgm/K

Material Name: BRONZE Bronze			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.16E+08 psi	0.11E+07 Kgf/cm/cm	0.11E+12 Pascals
NUXY	0.37	0.37	0.37
GXY	0.59E+07 psi	0.41E+06 Kgf/cm/cm	0.41E+11 Pascals
ALPX	0.18E-04 /Fahrenheit	0.32E-04 /Centigrade	0.32E-04 /Kelvin
DENS	0.81E-03 lbf s^2/in^4	0.88E-05 Kgf s^2/cm^4	0.87E+04 Kgm/m^3
KX	0.35E-03 BTU/in/s/F	0.62E-01 Cal/cm/s/C	26.0 W/m/K
C (Cp)	32.0 BTU in/lbf/s/s/F	0.81E+05 Cal cm/kgf/s/s/C	0.34E+03 J/kgm/K
Material Name: CA_STEEL Cast Alloy Steel (Below 8% Content)			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.28E+08 psi	0.20E+07 Kgf/cm/cm	0.19E+12 Pascals
NUXY	0.26	0.26	0.26
GXY	0.11E+08 psi	0.79E+06 Kgf/cm/cm	0.78E+11 Pascals
ALPX	0.81E-05 /Fahrenheit	0.15E-04 /Centigrade	0.15E-04 /Kelvin
DENS	0.68E-03 lbf s^2/in^4	0.74E-05 Kgf s^2/cm^4	0.73E+04 Kgm/m^3
KX	0.50E-03 BTU/in/s/F	0.90E-01 Cal/cm/s/C	38. W/m/K
C (Cp)	41.0 BTU in/lbf/s/s/F	0.10E+06 Cal cm/kgf/s/s/C	0.44E+03 J/kgm/K
Material Name: COBALT Cobalt			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.33E+08 psi	0.23E+07 Kgf/cm/cm	0.23E+12 Pascals
NUXY	0.31	0.31	0.31
GXY	0.13E+08 psi	0.90E+06 Kgf/cm/cm	0.88E+11 Pascals
ALPX	0.68E-05 /Fahrenheit	0.12E-04 /Centigrade	0.12E-04 /Kelvin
DENS	0.83E-03 lbf s^2/in^4	0.91E-05 Kgf s^2/cm^4	0.89E+04 Kgm/m^3
KX	0.92E-03 BTU/in/s/F	0.16 Cal/cm/s/C	69.0 W/m/K
C (Cp)	38.0 BTU in/lbf/s/s/F	0.96E+05 Cal cm/kgf/s/s/C	0.41E+03 J/kgm/K
Material Name: COPPER Copper			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.16E+08 psi	0.11E+07 Kgf/cm/cm	0.11E+12 Pascals
NUXY	0.37	0.37	0.37
GXY	0.58E+07 psi	0.41E+06 Kgf/cm/cm	0.40E+11 Pascals
ALPX	0.13E-04 /Fahrenheit	0.24E-04 /Centigrade	0.24E-04 /Kelvin
DENS	0.84E-03 lbf s^2/in^4	0.91E-05 Kgf s^2/cm^4	0.89E+04 Kgm/m^3
KX	0.52E-02 BTU/in/s/F	0.92 Cal/cm/s/C	0.39E+03 W/m/K
C (Cp)	36.0 BTU in/lbf/s/s/F	0.90E+05 Cal cm/kgf/s/s/C	0.39E+03 J/kgm/K
Material Name: CS_STEEL Cast Stainless Steel (Cf-8m or Cf-20)			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.28E+08 psi	0.20E+07 Kgf/cm/cm	0.19E+12 Pascals
NUXY	0.26	0.26	0.26
GXY	0.12E+08 psi	0.81E+06 Kgf/cm/cm	0.79E+11 Pascals
ALPX	0.84E-05 /Fahrenheit	0.15E-04 /Centigrade	0.15E-04 /Kelvin
DENS	0.73E-03 lbf s^2/in^4	0.79E-05 Kgf s^2/cm^4	0.77E+04 Kgm/m^3
KX	0.50E-03 BTU/in/s/F	0.90E-01 Cal/cm/s/C	37.0 W/m/K
C (Cp)	48.0 BTU in/lbf/s/s/F	0.12E+06 Cal cm/kgf/s/s/C	0.52E+03 J/kgm/K

Material Name: GC_IRON Gray Cast Iron (ASTM - Class 40)						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.17E+08	psi	0.12E+07	Kgf/cm/cm	0.12E+12	Pascals
NUXY	0.27		0.27		0.27	
GXY	0.72E+07	psi	0.51E+06	Kgf/cm/cm	0.50E+11	Pascals
ALPX	0.66E-05	/Fahrenheit	0.12E-04	/Centigrade	0.12E-04	/Kelvin
DENS	0.67E-03	lbf s^2/in^4	0.73E-05	Kgf s^2/cm^4	0.72E+04	Kgm/m^3
KX	0.60E-03	BTU/in/s/F	0.11	Cal/cm/s/C	45.0	W/m/K
C (Cp)	47.0	BTU in/lbf/s/s/F	0.12E+06	Cal cm/kgf/s/s/C	0.51E+03	J/kgm/K
Material Name: GLASS Glass						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.92E+07	psi	0.64E+06	Kgf/cm/cm	0.63E+11	Pascals
NUXY	0.22		0.22		0.22	
GXY	0.15E+08	psi	0.10E+07	Kgf/cm/cm	0.10E+12	Pascals
ALPX	0.55E-05	/Fahrenheit	0.99E-05	/Centigrade	0.99E-05	/Kelvin
DENS	0.28E-03	lbf s^2/in^4	0.31E-05	Kgf s^2/cm^4	0.30E+04	Kgm/m^3
KX	0.22E-04	BTU/in/s/F	0.38E-02	Cal/cm/s/C	1.6	W/m/K
C (Cp)	70.0	BTU in/lbf/s/s/F	0.18E+06	Cal cm/kgf/s/s/C	0.75E+03	J/kgm/K
Material Name: GOLD Pure Gold						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.11E+08	psi	0.76E+06	Kgf/cm/cm	0.74E+11	Pascals
NUXY	0.42		0.42		0.42	
GXY	0.38E+07	psi	0.27E+06	Kgf/cm/cm	0.26E+11	Pascals
ALPX	0.79E-05	/Fahrenheit	0.14E-04	/Centigrade	0.14E-04	/Kelvin
DENS	0.18E-02	lbf s^2/in^4	0.20E-04	Kgf s^2/cm^4	0.19E+05	Kgm/m^3
KX	0.40E-02	BTU/in/s/F	0.71	Cal/cm/s/C	0.30E+03	W/m/K
C (Cp)	12.0	BTU in/lbf/s/s/F	0.30E+05	Cal cm/kgf/s/s/C	0.13E+03	J/kgm/K
Material Name: IRON Iron						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.17E+08	psi	0.12E+07	Kgf/cm/cm	0.12E+12	Pascals
NUXY	0.31		0.31		0.31	
GXY	0.11E+08	psi	0.78E+06	Kgf/cm/cm	0.77E+11	Pascals
ALPX	0.61E-05	/Fahrenheit	0.11E-04	/Centigrade	0.11E-04	/Kelvin
DENS	0.74E-03	lbf s^2/in^4	0.80E-05	Kgf s^2/cm^4	0.79E+04	Kgm/m^3
KX	0.10E-02	BTU/in/s/F	0.18	Cal/cm/s/C	75.	W/m/K
C (Cp)	42.0	BTU in/lbf/s/s/F	0.11E+06	Cal cm/kgf/s/s/C	0.45E+03	J/kgm/K
Material Name: LEAD Pure Lead						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.20E+07	psi	0.14E+06	Kgf/cm/cm	0.14E+11	Pascals
NUXY	0.40		0.40		0.40	
GXY	0.71E+06	psi	0.50E+05	Kgf/cm/cm	0.49E+10	Pascals
ALPX	0.29E-04	/Fahrenheit	0.53E-04	/Centigrade	0.53E-04	/Kelvin
DENS	0.11E-02	lbf s^2/in^4	0.12E-04	Kgf s^2/cm^4	0.11E+05	Kgm/m^3
KX	0.47E-03	BTU/in/s/F	0.83E-01	Cal/cm/s/C	35.	W/m/K
C (Cp)	12.0	BTU in/lbf/s/s/F	0.30E+05	Cal cm/kgf/s/s/C	0.13E+03	J/kgm/K

Material Name: MAGNES Magnesium Alloy - Wrought or Cast			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.65E+07 psi	0.46E+06 Kgf/cm/cm	0.45E+11 Pascals
NUXY	0.35	0.35	0.35
GXY	0.24E+07 psi	0.17E+06 Kgf/cm/cm	0.17E+11 Pascals
ALPX	0.14E-04 /Fahrenheit	0.25E-04 /Centigrade	0.25E-04 /Kelvin
DENS	0.16E-03 $lbf\,s^2/in^4$	0.18E-05 $Kgf\,s^2/cm^4$	0.17E+04 Kgm/m^3
KX	0.21E-02 BTU/in/s/F	0.38 Cal/cm/s/C	0.16E+03 W/m/K
C (Cp)	97.0 BTU in/lbf/s/s/F	0.25E+06 Cal cm/kgf/s/s/C	0.10E+04 J/kgm/K
Material Name: MC_IRON Malleable Cast Iron (ASTM - A220)			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.27E+08 psi	0.19E+07 Kgf/cm/cm	0.19E+12 Pascals
NUXY	0.27	0.27	0.27
GXY	0.13E+08 psi	0.88E+06 Kgf/cm/cm	0.86E+11 Pascals
ALPX	0.67E-05 /Fahrenheit	0.12E-04 /Centigrade	0.12E-04 /Kelvin
DENS	0.68E-03 $lbf\,s^2/in^4$	0.74E-05 $Kgf\,s^2/cm^4$	0.73E+04 Kgm/m^3
KX	0.63E-03 BTU/in/s/F	0.11 Cal/cm/s/C	47.0 W/m/K
C (Cp)	47.0 BTU in/lbf/s/s/F	0.12E+06 Cal cm/kgf/s/s/C	0.51E+03 J/kgm/K
Material Name: MN_IRON Maganese Bronze			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.15E+08 psi	0.11E+07 Kgf/cm/cm	0.11E+12 Pascals
NUXY	0.37	0.37	0.37
GXY	0.56E+07 psi	0.39E+06 Kgf/cm/cm	0.39E+11 Pascals
ALPX	0.12E-04 /Fahrenheit	0.22E-04 /Centigrade	0.22E-04 /Kelvin
DENS	0.78E-03 $lbf\,s^2/in^4$	0.85E-05 $Kgf\,s^2/cm^4$	0.83E+04 Kgm/m^3
KX	0.13E-03 BTU/in/s/F	0.23E-01 Cal/cm/s/C	9.6 W/m/K
C (Cp))	35.0 BTU in/lbf/s/s/F	0.88E+05 Cal cm/kgf/s/s/C	0.38E+03 J/kgm/K
Material Name: MOLYBDENUM Molybdenum			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.47E+08 psi	0.33E+07 Kgf/cm/cm	0.32E+12 Pascals
NUXY	0.38	0.38	0.38
GXY	0.17E+08 psi	0.12E+07 Kgf/cm/cm	0.12E+12 Pascals
ALPX	0.28E-05 /Fahrenheit	0.50E-05 /Centigrade	0.50E-05 /Kelvin
DENS	0.96E-03 $lbf\,s^2/in4$	0.10E-04 $Kgf\,s^2/cm4$	0.10E+05 Kgm/m^3
KX	0.20E-02 BTU/in/s/F	0.35 Cal/cm/s/C	0.15E+03 W/m/K
C (Cp)	25.0 BTU in/lbf/s/s/F	0.64E+05 Cal cm/kgf/s/s/C	0.27E+03 J/kgm/K
Material Name: MONEL Monel 400 Alloy of Nickel			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.26E+08 psi	0.18E+07 Kgf/cm/cm	0.18E+12 Pascals
NUXY	0.34	0.34	0.34
GXY	0.97E+07 psi	0.68E+06 Kgf/cm/cm	0.67E+11 Pascals
ALPX	0.77E-05 /Fahrenheit	0.14E-04 /Centigrade	0.14E-04 /Kelvin
DENS	0.83E-03 $lbf\,s^2/in^4$	0.90E-05 $Kgf\,vs^2/cm^4$	0.88E+04 Kgm/m^3
KX	0.29E-03 BTU/in/s/F	0.52E-01 Cal/cm/s/C	22.0 W/m/K
C (Cp)	39.0 BTU in/lbf/s/s/F	0.10E+06 Cal vcm/kgf/s/s/C	0.43E+03 J/kgm/K

Material Name: NICKEL Nickel						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.31E+08	psi	0.21E+07	Kgf/cm/cm	0.21E+12	Pascals
NUXY	0.31		0.31		0.31	
GXY	0.11E+08	psi	0.81E+06	Kgf/cm/cm	0.79E+11	Pascals
ALPX	0.92E-05	/Fahrenheit	0.17E-04	/Centigrade	0.17E-04	/Kelvin
DENS	0.80E-03	lbf s^2/in^4	0.87E-05	Kgf s^2/cm^4	0.85E+04	Kgm/m^3
KX	0.58E-03	BTU/in/s/F	0.10	Cal/cm/s/C	43.	W/m/K
C (Cp)	42.0	BTU in/lbf/s/s/F	0.11E+06	Cal cm/kgf/s/s/C	0.46E+03	J/kgm/K
Material Name: D_NICKEL Duranickel 301						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.30E+08	psi	0.21E+07	Kgf/cm/cm	0.21E+12	Pascals
NUXY	0.34		0.34		0.34	
GXY	0.11E+08	psi	0.79E+06	Kgf/cm/cm	0.77E+11	Pascals
ALPX	0.82E-05	/Fahrenheit	0.15E-04	/Centigrade	0.15E-04	/Kelvin
DENS	0.77E-03	lbf s^2/in^4	0.84E-05	Kgf s^2/cm^4	0.82E+04	Kgm/m^3
KX	0.32E-03	BTU/in/s/F	0.57E-01	Cal/cm/s/C	24.0	W/m/K
C (Cp)	54.0	BTU in/lbf/s/s/F	0.14E+06	Cal cm/kgf/s/s/C	0.59E+03	J/kgm/K
Material Name: NYLON Nylon 6/10						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.12E+07	psi	0.84E+05	Kgf/cm/cm	0.83E+10	Pascals
NUXY	0.28		0.28		0.28	
GXY	0.47E+06	psi	0.33E+05	Kgf/cm/cm	0.32E+10	Pascals
ALPX	0.16E-04	/Fahrenheit	0.30E-04	/Centigrade	0.30E-04	/Kelvin
DENS	0.13E-03	lbf s^2/in^4	0.14E-05	Kgf s^2/cm^4	0.14E+04	Kgm/m^3
KX	0.70E-05	BTU/in/s/F	0.13E-02	Cal/cm/s/C	0.53	W/m/K
C (Cp)	0.14E+03	BTU in/lbf/s/s/F	0.34E+06	Cal cm/kgf/s/s/C	0.15E+04	J/kgm/K
Material Name: PC_STEEL Plain Carbon Steel						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.30E+08	psi	0.21E+07	Kgf/cm/cm	0.21E+12	Pascals
NUXY	0.28		0.28		0.28	
GXY	0.12E+08	psi	0.81E+06	Kgf/cm/cm	0.79E+11	Pascals
ALPX	0.74E-05	/Fahrenheit	0.13E-04	/Centigrade	0.13E-04	/Kelvin
DENS	0.73E-03	lbf s^2/in^4	0.79E-05	Kgf s^2/cm^4	0.78E+04	Kgm/m^3
KX	0.57E-03	BTU/in/s/F	0.10	Cal/cm/s/C	43.	W/m/K
C (Cp)	41.0	BTU in/lbf/s/s/F	0.10E+06	Cal cm/kgf/s/s/C	0.44E+03	J/kgm/K
Material Name: PORCELAIN Ceramic Porcelain						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.15E+08	psi	0.11E+07	Kgf/cm/cm	0.10E+12	Pascals
NUXY	0.22		0.22		0.22	
GXY	0.15E+08	psi	0.10E+07	Kgf/cm/cm	0.10E+12	Pascals
ALPX	0.35E-05	/Fahrenheit	0.63E-05	/Centigrade	0.63E-05	/Kelvin
DENS	0.24E-03	lbf s^2/in^4	0.26E-05	Kgf s^2/cm^4	0.25E+04	Kgm/m^3
KX	0.68E-04	BTU/in/s/F	0.12E-01	Cal/cm/s/C	5.0	W/m/K
C (Cp)	81.0	BTU in/lbf/s/s/F	0.21E+06	Cal cm/kgf/s/s/C	0.88E+03	J/kgm/K

Material Name: RUBBER Rubber							
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.88E+03	psi	62.	Kgf/cm/cm	0.61E+07	Pascals
NUXY	0.4		0.49		0.49	
GXY	0.42E+03	psi	29.0	Kgf/cm/cm	0.29E+07	Pascals
ALPX	0.37E-03	/Fahrenheit	0.67E-03	/Centigrade	0.67E-03	/Kelvin
DENS	0.93E-04	$lbf\,s^2/in^4$	0.10E-05	$Kgf\,s^2/cm^4$	0.10E+04	Kgm/m^3
KX	0.19E-05	BTU/in/s/F	0.34E-03	Cal/cm/s/C	0.14	W/m/K
Material Name: SILVER Pure Silver						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.10E+08	psi	0.70E+06	Kgf/cm/cm	0.69E+11	Pascals
NUXY	0.37		0.37		0.37	
GXY	0.36E+07	psi	0.26E+06	Kgf/cm/cm	0.25E+11	Pascals
ALPX	0.11E-04	/Fahrenheit	0.20E-04	/Centigrade	0.20E-04	/Kelvin
DENS	0.98E-03	$lbf\,s^2/in^4$	0.11E-04	$Kgf\,s^2/cm^4$	0.11E+05	Kgm/m^3
KX	0.56E-02	BTU/in/s/F	1.0	Cal/cm/s/C	0.42E+03	W/m/K
C (Cp)	22.0	BTU in/lbf/s/s/F	0.55E+05	Cal cm/kgf/s/s/C	0.23E+03	J/kgm/K
Material Name: STEEL Steel						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.29E+08	psi	0.20E+07	Kgf/cm/cm	0.20E+12	Pascals
NUXY	0.32		0.32		0.32	
GXY	0.11E+08	psi	0.77E+06	Kgf/cm/cm	0.76E+11	Pascals
ALPX	0.65E-05	/Fahrenheit	0.12E-04	/Centigrade	0.12E-04	/Kelvin
DENS	0.73E-03	$lbf\,s^2/in^{**4}$	0.80E-05	$Kgf\,s^2/cm^{**4}$	0.78E+04	Kgm/m^3
KX	0.40E-03	BTU/in/s/F	0.72E-01	Cal/cm/s/C	30.0	W/m/K
C (Cp)	46.0	BTU in/lbf/s/s/F	0.12E+06	Cal cm/kgf/s/s/C	0.50E+03	J/kgm/K
Material Name: ST_1020 Steel, AISI C1020 (Hot Worked)						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.29E+08	psi	0.20E+07	Kgf/cm/cm	0.20E+12	Pascals
NUXY	0.29		0.29		0.29	
GXY	0.11E+08	psi	0.79E+06	Kgf/cm/cm	0.77E+11	Pascals
ALPX	0.84E-05	/Fahrenheit	0.15E-04	/Centigrade	0.15E-04	/Kelvin
DENS	0.74E-03	$lbf\,s^2/in^4$	0.80E-05	$Kgf\,s^2/cm^4$	0.79E+04	Kgm/m^3
KX	0.62E-03	BTU/in/s/F	0.11	Cal/cm/s/C	47.0	W/m/K
C (Cp)	39.0	BTU in/lbf/s/s/F	0.98E+05	Cal cm/kgf/s/s/C	0.42E+03	J/kgm/K
Material Name: ST_304 Steel, AISI 304 (Sheet)						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.28E+08	psi	0.20E+07	Kgf/cm/cm	0.19E+12	Pascals
NUXY	0.29		0.29		0.29	
GXY	0.11E+08	psi	0.76E+06	Kgf/cm/cm	0.75E+11	Pascals
ALPX	0.99E-05	/Fahrenheit	0.18E-04	/Centigrade	0.18E-04	/Kelvin
DENS	0.75E-03	$lbf\,s^2/in^4$	0.82E-05	$Kgf\,s^2/cm^4$	0.80E+04	Kgm/m^3
KX	0.22E-03	BTU/in/s/F	0.39E-01	Cal/cm/s/C	16.0	W/m/K
C (Cp)	46.0	BTU in/lbf/s/s/F	0.12E+06	Cal cm/kgf/s/s/C	0.50E+03	J/kgm/K

Material Name: ST_ST Stainless Steel						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.29E+08	psi	0.20E+07	Kgf/cm/cm	0.20E+12	Pascals
NUXY	0.28		0.28		0.28	
GXY	0.11E+08	psi	0.78E+06	Kgf/cm/cm	0.77E+11	Pascals
ALPX	0.60E-05	/Fahrenheit	0.11E-04	/Centigrade	0.11E-04	/Kelvin
DENS	0.73E-03	lbf s^2/in^4	0.80E-05	Kgf s^2/cm^4	0.78E+04	Kgm/m^3
KX	0.24E-03	BTU/in/s/F	0.43E-01	Cal/cm/s/C	18.0	W/m/K
C (Cp)	42.0	BTU in/lbf/s/s/F	0.11E+06	Cal cm/kgf/s/s/C	0.46E+03	J/kgm/K
Material Name: T_BRONZE Tin Brearing Bronze						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.13E+08	psi	0.91E+06	Kgf/cm/cm	0.90E+11	Pascals
NUXY	0.33		0.33		0.33	
GXY	0.54E+07	psi	0.38E+06	Kgf/cm/cm	0.37E+11	Pascals
ALPX	0.10E-04	/Fahrenheit	0.18E-04	/Centigrade	0.18E-04	/Kelvin
DENS	0.78E-03	lbf s^2/in^4	0.85E-05	Kgf s^2/cm^4	0.83E+04	Kgm/m^3
KX	0.62E-03	BTU/in/s/F	0.11	Cal/cm/s/C	47.0	W/m/K
C (Cp)	35.0	BTU in/lbf/s/s/F	0.88E+05	Cal cm/kgf/s/s/C	0.38E+03	J/kgm/K
Material Name: TITANIUM Titanium						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.16E+08	psi	0.11E+07	Kgf/cm/cm	0.11E+12	Pascals
NUXY	0.30		0.3		0.30	
GXY	0.63E+07	psi	0.44E+06	Kgf/cm/cm	0.43E+11	Pascals
ALPX	0.49E-05	/Fahrenheit	0.88E-05	/Centigrade	0.88E-05	/Kelvin
DENS	0.43E-03	lbf s^2/in^4	0.47E-05	Kgf s^2/cm^4	0.46E+04	Kgm/m^3
KX	0.29E-03	BTU/in/s/F	0.52E-01	Cal/cm/s/C	22.0	W/m/K
C (Cp)	2.0	BTU in/lbf/s/s/F	0.11E+06	Cal cm/kgf/s/s/C	0.46E+03	J/kgm/K
Material Name: TUNGSTEN Tungsten						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.59E+08	psi	0.41E+07	Kgf/cm/cm	0.41E+12	Pascals
NUXY	0.28		0.28		0.28	
GXY	0.23E+08	psi	0.16E+07	Kgf/cm/cm	0.16E+12	Pascals
ALPX	0.25E-05	/Fahrenheit	0.45E-05	/Centigrade	0.45E-05	/Kelvin
DENS	0.18E-02	lbf s^2/in^4	0.20E-04	Kgf s^2/cm^4	0.19E+05	Kgm/m^3
KX	0.27E-02	BTU/in/s/F	0.48	Cal/cm/s/C	0.20E+03	W/m/K
C (Cp)	12.0	BTU in/lbf/s/s/F	0.31E+05	Cal cm/kgf/s/s/C	0.13E+03	J/kgm/K
Material Name: VANADIUM Vanadium						
Property Name	**Value (English)**		**Value (MKS)**		**Value (SI)**	
EX	0.20E+08	psi	0.14E+07	Kgf/cm/cm	0.14E+12	Pascals
NUXY	0.36		0.36		0.36	
GXY	0.74E+07	psi	0.52E+06	Kgf/cm/cm	0.51E+11	Pascals
ALPX	0.46E-05	/Fahrenheit	0.83E-05	/Centigrade	0.83E-05	/Kelvin
DENS	0.57E-03	lbf s^2/in^4	0.62E-05	Kgf s^2/cm^4	0.61E+04	Kgm/m^3
KX	0.41E-03	BTU/in/s/F	0.74E-01	Cal/cm/s/C	31.0	W/m/K
C (Cp)	46.0	BTU in/lbf/s/s/F	0.12E+06	Cal cm/kgf/s/s/C	0.50E+03	J/kgm/K

Material Name: W_COPPER Wrought Cooper			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.16E+08 psi	0.11E+07 Kgf/cm/cm	0.11E+12 Pascals
NUXY	0.33	0.33	0.33
GXY	0.58E+07 psi	0.41E+06 Kgf/cm/cm	0.40E+11 Pascals
ALPX	0.11E-04 /Fahrenheit	0.20E-04 /Centigrade	0.20E-04 /Kelvin
DENS	0.79E-03 lbf s^2/in^4	0.86E-05 Kgf s^2/cm^4	0.84E+04 Kgm/m^3
KX	0.31E-02 BTU/in/s/F	0.56 Cal/cm/s/C	0.23E+03 W/m/K
C (Cp)	35.0 BTU in/lbf/s/s/F	0.88E+05 Cal cm/kgf/s/s/C	0.38E+03 J/kgm/K
Material Name: WS_STEEL Wrought Stainless Steel			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.29E+08 psi	0.20E+07 Kgf/cm/cm	0.20E+12 Pascals
NUXY	0.26	0.26	0.26
GXY	0.12E+08 psi	0.81E+06 Kgf/cm/cm	0.79E+11 Pascals
ALPX	0.60E-05 /Fahrenheit	0.11E-04 /Centigrade	0.11E-04 /Kelvin
DENS	0.75E-03 lbfs^2/in^4	0.82E-05 Kgf s^2/cm^4	0.80E+04 Kgm/m^3
KX	0.25E-03 BTU/in/s/F	0.45E-01 Cal/cm/s/C	19.0 W/m/K
C (Cp)	46.0 BTU in/lbf/s/s/F	0.12E+06 Cal cm/kgf/s/s/C	0.50E+03 J/kgm/K
Material Name: ZIRCONIUM Zirconium			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
EX	0.14E+08 psi	0.10E+07 Kgf/cm/cm	0.98E+11 Pascals
NUXY	0.34	0.34	0.34
GXY	0.53E+07 psi	0.37E+06 Kgf/cm/cm	0.36E+11 Pascals
ALPX	0.61E-05 /Fahrenheit	0.11E-04 /Centigrade	0.11E-04 /Kelvin
DENS	0.62E-03 lbf s^2/in^4	0.67E-05 Kgf s^2/cm^4	0.66E+04 Kgm/m^3
KX	0.22E-04 BTU/in/s/F	0.40E-02 Cal/cm/s/C	1.7 W/m/K
C (Cp)	27.0 BTU in/lbf/s/s/F	0.69E+05 Cal cm/kgf/s/s/C	0.29E+03 J/kgm/K
Material Name: WATER Water			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
DENS	0.93E-04 lbf s^2/in^4	0.10E-05 Kgf*s^2/cm*	0.10E+04 Kgm/m^3
KX	0.82E-05 BTU/in/s/F	0.15E-02 Cal/cm/s/C	0.61 W/m/K
C (Cp)	0.39E+03 BTU in/lbf/s/s/F	0.98E+06 Cal cm/kgf/s/s/C	0.42E+04 J/kgm/K
VISC	0.13E-06 lbf s/in/in	0.88E-08 kgf s/cm/cm	0.87E-03 kgm/m/s
BETA	0.15E-03 /Rankin	0.27E-03 /Kelvin	0.27E-03 /Kelvin
Material Name: AIR Air (300 K)			
Property Name	**Value (English)**	**Value (MKS)**	**Value (SI)**
DENS	0.10E-06 lbf s^2/in^4	0.11E-08 Kgf s^2/cm^4	1.1Kgm/m^3
KX	0.36E-06 BTU/in/s/F	0.64E-04 Cal/cm/s/C	0.27E-01 W/m/K
C (Cp)	93.0 BTU in/lbf/s/s/F	0.24E+06 Cal cm/kgf/s/s/C	0.10E+04 J/kgm/K
VISC	0.27E-08 lbf s/in/in	0.19E-09 kgf s/cm/cm	0.19E-04 kgm/m/s
BETA	0.19E-02 /Rankin	0.33E-02 /Kelvin	0.33E-02 /Kelvin
GAMMA	1.40	1.40	1.40

Using Your Own Material Library

If you need to frequently use material properties not available in the built-in material library, COSMOS/M provides you the feature to add material properties to a user-created library. The **USER_MAT** (Propsets > **User Material Lib**) command allows you to read your own material properties defined in the file USERMAT.LIB which has the same format as the PICKMAT.LIB file. Note that the file, USERMAT.LIB, has already been installed in the COSMOS/M directory, and its contents are the same as those in the PICKMAT.LIB file. Currently, there is *no* limit on the number of materials for which you can pre-define material properties in the USERMAT.LIB file.

The format of a typical material property entry existing in the USERMAT.LIB file is shown below. To define your own material properties, you need to modify the material names and the numerical values *without* altering the existing format.

```
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MATL:W_COPPER :   WROUGHT COPPER
EX        15.5E+6  p.s.i.          1.0897E6  kg/sq.cm.        106.87E+9 Pascals
NUXY      0.33                     0.33                       0.33
GXY       5.80E+6  p.s.i.          4.078E+5  kg/sq.cm.        39.99E+9  Pascals
ALPX      11.0E-6  /Fahrenheit     19.80E-6  /Centigrade      19.80E-6  /Kelvin
DENS      7.893E-4 lb.s.s/in**4    8.6059E-6 kgf.s.s/cm**4    8442.4    kg/cu.m
KX        5.370E-3 BTU/in.s.F      0.062     Cal/cm.s.C       26        W/m.K
C (Cp)    34.776   BTUin/lbssF     88305.696 Calcm/kgssC      376.64    J/kg.K
ENDMATL
```

As you can notice from the above, properties for each material are defined starting with a `MATL`: statement and are terminated with the `ENDMATL` statement. The format of entries in between these statements should be identical to the entries shown above. For example, if you like to define material properties for Concrete (in compression *only*) in FPS units, the changes to the above lines are as follows:

```
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MATL:CONCRETE :    CEMENT CONCRETE
EX        3.0E+6   p.s.i.          *.******  kg/sq.cm.        ***.****  Pascals
NUXY      0.18                     0.18                       0.18
GXY       1.27E+6  p.s.i.          *.******  kg/sq.cm.        **.*****  Pascals
ALPX      1.25E-5  /Fahrenheit     *.******  /Centigrade      **.*****  /Kelvin
DENS      2.20E-4  lb.s.s/in**4    *.******* kgf.s.s/cm**4    ******    kg/cu.m
KX        0.2      BTU/in.s.F      *.***     Cal/cm.s.C       **        W/m.K
C (Cp)    7.2E-4   BTUin/lbssF     *****.*** Calcm/kgssC      ***.**    J/kg.K
ENDMATL
```

You can complete the remaining entries by suitably multiplying the values in FPS units by conversion factors (see Appendix A, *Units*, for conversion factors). In order to use the properties defined above, the **USER_MAT** (Propsets > **User Material Lib**) command for material property set 1 in FPS units is executed as follows:

Geo Panel: Propsets > **User Material Lib**
Material Property set > **1**
Material Name > **CONCRETE**
Unit Label > **FPS**

Modification, Display, and Output of Material Property Definitions

The command **MPDEL** (Edit > DELETE > **Material Props**) can be used to delete *all* properties defined in one or more material sets. If you like to delete or modify a specific property in a material set, then you need to execute the **MPROP** (Propsets > **Material Property**) command, specify the appropriate material set number and the property name, and simply enter the new value for that property.

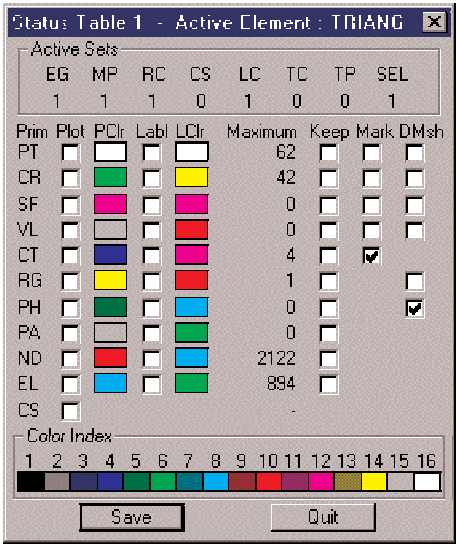
During mesh generation, the material group currently active is associated with the elements created. If you have more than one material group in your model and would like to make a particular group active, then you need to use the command **ACTSET** (Control > ACTIVATE > **Set Entity**). In addition to activating material sets, this command can be used to activate sets for real constants, coordinate systems, temperature curves, and many others. The description of these sets and their set names are shown in the table below.

Table 6-10. Description of Set Names for Activating Using ACTSET Command

Set Name	Description	Set Name	Description
EG	Element group	BH	B-H curve (for magnetic analysis)
MP	Material property set	CS	Coordinate system
RC	Real constant set	ECS	Element coordinate system
LC	Load case		
TC	Time curve	LOC	Fatigue location
TP	Temperature curve	MC	Material curve

The set names that are currently active can be graphically viewed on the screen using the **STATUS1** icon as shown below.

Figure 6-4. Status Table for Sets and Entities

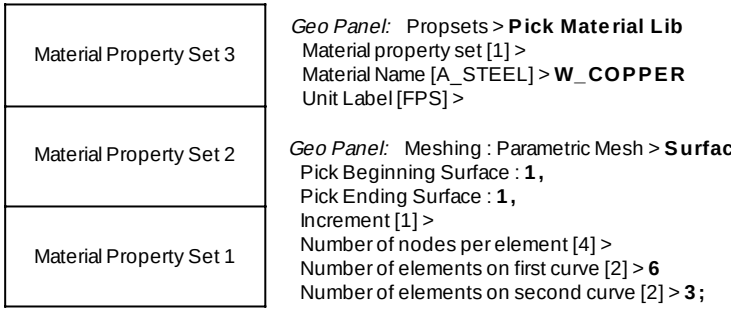


To change the association of *existing* material property sets with element groups and real constant sets, the **EPROPCHANGE** (Propsets > **Change El-Prop**) command can be used. This command also assigns a specified color to a selected pattern of elements. The assigned colors can be subsequently used to display and hence identify parts of your mesh with different material properties, element groups and real constant sets. The next section explains this in more detail.

Display of Material Property Definitions

For models with more than one material group, COSMOS/M provides a novel way to graphically represent the different material groups for model checking purposes. The command **ACTECLR** (Meshing > ELEMENTS > **Activate Elem Color**) can be used to activate the *default* colors for elements and one of the associated sets (the material set in this case). The following demonstration explains the procedure for a 2D model with three different material properties as shown in the figure below.

Figure 6-5. Example with Multiple Material Sets for Element Coloring



To start with, construct three connected surfaces as shown in the figure. Define the material properties for the first surface using either the **MPROP** (Propsets > **Material Property**) or **PICK_MAT** (Propsets > **Pick Material Lib**) (illustrated above) command. Next, generate the finite element mesh for the first surface so that the elements generated will be associated with material set 1. Before moving on to surface 2, define the material properties for set 2, and then proceed to meshing surface 2. Repeat this procedure for the last surface. If you list the elements generated using the **ELIST** (Edit > LIST > **Elements**) command, you will observe that the elements generated for each surface show the respective association of material sets.

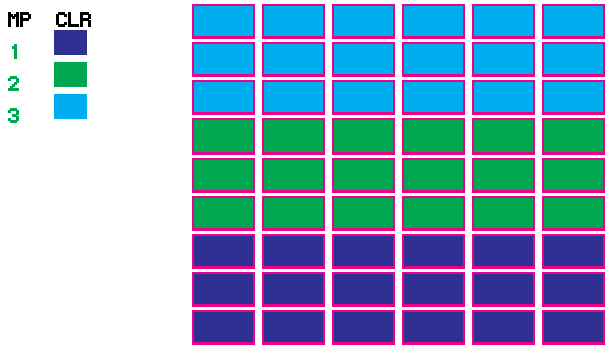
To display the finite element mesh with default colors for material properties, you need to use the command **ACTECLR** (Meshing > ELEMENTS > **Activate Elem Color**), as illustrated below:

Geo Panel: Meshing > ELEMENTS > **Activate Elem Color**
Color Flag > **Activate**

Set Label > **Material Property Set**
Default colors flag > **Yes**

The above procedure results in default colors (use the **STATUS1** icon to see GEOSTAR's default colors and their numbers) assigned to elements with different material property set numbers. Next, use the command **EPLOT** (Edit > PLOT > **Elements**) to plot the elements with different material sets. To obtain a more clear view, you can shrink and shade the elements using the commands **SHRINK** (Display > DISPLAY OPTION > **Shrink**) and **SHADE** (Display > DISPLAY OPTION > **Shaded Element Plot**). The figure below shows the *shrunken* finite element mesh with default coloring for material sets.

Figure 6-6. Element Shrink Plot with Default Colors for Material Groups



If you want to assign colors other than the default for elements, you need to use the command **SETECLR** (Meshing > ELEMENTS > **Set Element Color**). You need to re-execute this command separately for each material set number and assign the desired color.

Material Property Output

By default, the material property information is *not* written in the output file, jobname.OUT, created after you successfully complete the analysis. If you want

to include material property information, you need to request for a detailed output by activating the input print flag using the command **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**); see illustration below:

Geo Panel: Analysis > OUTPUT INTERVAL OPTIONS > **Set Print Options**
Displacement Print Interval > **Yes**
... ...
Input Print Flag > **Yes**

The material properties defined can also be listed on the screen using the **MPLIST** (Edit > LIST > **Material Props**) command. This command will list material properties defined for all or specified material groups and will also indicate the material set currently active (alphabet A), as shown below:

	Label	Name	Temp/BH_Cr	Value
	1	EX	0	1.550000e+07
	1	NUXY	0	3.300000e-01
	1	GXY	0	5.800000e+06
	1	ALPX	0	1.100000e-05
	1	DENS	0	7.893000e-04
	1	C	0	3.477600e+01
	1	KX	0	5.370000e-03
A	2	EX	0	1.550000e+07
A	2	NUXY	0	3.300000e-01
A	2	GXY	0	5.800000e+06
A	2	ALPX	0	1.100000e-05
A	2	DENS	0	7.893000e-04
A	2	C	0	3.477600e+01
A	2	KX	0	5.370000e-03

As with all listing commands in GEOSTAR, the information from **MPLIST** (Edit > LIST > **Material Props**) command can be *routed* to an ascii file using the command **LISTLOG** (Control > MISCELLANEOUS > **List Log**).




Sectional and Other Physical Properties

In COSMOS/M, the sectional properties of elements and related physical properties *not* covered under the **MPROP** (Propsets > **Material Property**) command are called *real constants* and are defined using the **RCONST** (Propsets > **Real Constant**) command. The term real constant refers to various sectional properties such as area, width and depth or thickness of the cross section, perimeter, moment of inertia, radius of curvature, internal hinges, material angles for orthotropic materials, temperature gradients, spring constants, layer information for laminated composites, and many other physical properties (see Table 6-11 for a list of real constants for all elements in the Basic FEA System) that are necessary to compute the element stiffness matrices. The term has nothing to do with the nature of the numeric data you input, whether real or integer. Since the sectional and related physical constants depend on the type of element you are using, the element group information ***must*** be defined prior to real constants definition. The **RCONST** (Propsets > **Real Constant**) command defines a real constant set number, associates this with the an element group number, and then prompts you for the *required* real constants for the specified element type. The prompts for the **RCONST** (Propsets > **Real Constant**) command therefore depend on the type of element you are using, and they usually provide sufficient physical interpretation so that you can enter the required input with minimal difficulty. The use of **RCONST** (Propsets > **Real Constant**) command requires you to provide the element group number associated with the real constant set, and the real constant set number. The argument on starting location of the first constant is fully explained in the next section.

✍ The maximum number of real constant sets permitted in a model is 5000.

As shown in Table 6-11, the number of real constants to be input varies from a small number such as 1 (for a rigid bar element) to as high as 78 (for a general stiffness element). However, the *maximum* number of real constants you can input using **RCONST** (Propsets > **Real Constant**) in *one* attempt is 10. If you need to input more constants, you need to re-execute the **RCONST** (Propsets > **Real Constant**) command, and input the remaining constants. In this case, the program will automatically prompt you with a starting location of 11 and onwards. If you make a mistake in entering the section constants, you do not need to redefine all of the section constants. You can move to a specific location (using the starting location concept, explained in the next section) of the real constants and input only the erroneous values. The program will also warn you if you input fewer than the required number of constants.




Similar to the material library feature, COSMOS/M also provides built-in section libraries for standard cross sections. The command **PICK_SEC** (Propsets > **AISC Sect Table**) selects standard steel sections and the associated section properties from AISC (American Institute of Steel Construction) tables while the command **BMSECDEF** (Propsets > **Beam Section**) will allow you to select standard as well as user-defined sections from the COSMOS/M section library. When you use these commands, you will also be assigning a real constant set number, but the section properties are internally computed by the program. The command **BMSECLIST** (Edit > LIST > **Beam Section**) can be used to list the section selected from the COSMOS/M section library.

The command **RCDEL** (Edit > DELETE > **Real Constants**) deletes the section property set number and the associated properties defined using **RCONST** (Propsets > **Real Constant**), **PICK_SEC** (Propsets > **AISC Sect Table**), or **BMSECDEF** (Propsets > **Beam Section**) commands. The command **RCLIST** (Edit > LIST > **Real Constants**) can be used to list the section properties defined on the screen. For more information on the modification of section property definitions, please refer to the section, *Modification, Display, and Output of Section Property Definitions*.

To illustrate the input of section properties using the **RCONST** (Propsets > **Real Constant**) command, consider an example using a 3-node composite shell element (SHELL3L) with four layers (defined using **EGROUP** (Propsets > **Element Group**) command). In COSMOS/M, the layer data such as thickness and orientation of material principal axes are entered as real constants. For each layer, you will be prompted for the layer thickness, material set number, and material angle. Referring to Table 6-11, for four layers, there will be 14 section constants required to be input. As mentioned before, the **RCONST** (Propsets > **Real Constant**) command accepts a maximum of 10 input data in each execution, and you therefore need to re-execute this command to input data for the remaining layers as illustrated below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **10**

RC1 : Dist. from ref. plane to top surface > **0.5**
RC2 : Temperature gradient > **0**
RC3 : Thickness of layer 1 > **0.25**
RC4 : Material number for layer 1 > **1**
RC5 : Material angle for layer 1 > **0**
RC6 : Thickness of layer 2 > **0.25**
RC7 : Material number for layer 2 > **1**
RC8 : Material angle for layer 2 > **30**
RC9 : Thickness of layer 3 > **0.25**
RC10 : Material number for layer 3 > **1**

Re-execute the **RCONST** (Propsets > **Real Constant**) command for the remaining data, as shown below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **11**
No. of real constants to be entered > **4**

RC11 : Material angle for layer 3 > **60**
RC12 : Thickness of layer 4 > **0.25**
RC13 : Material number for layer 4 > **1**
RC14 : Material angle for layer 4 > **90**

Note that in order to continue entering the section constants, you need to specify the same real constant set number.

Table 6-11 in the following pages outlines the section constant information required for all elements in the Basic FEA System. The concept of starting locations is explained more clearly by means of an example in the next section, followed by detailed descriptions of real constant data for all elements in the Basic FEA System. The descriptions are presented separately for 1D elements (including truss and beam), 2D continuum elements, and 3D solid elements. The real constant data for rigid bar, gap, spring, mass, boundary and general stiffness elements is presented separately under special elements category.




Table 6-11. Section Constants for All Elements in the Basic FEA System

COSMOS/M Element Name	Section Property Constants Required		
	Number	RC	Description
TRUSS2D	2	RC1 RC2	Cross sectional area Cross sectional perimeter (for thermal analysis only)
TRUSS3D	2	RC1 RC2	Cross sectional area Cross sectional perimeter (for thermal analysis only)
BEAM2D	8	RC1 RC2 RC3 RC4 RC5 RC6 RC7 RC8	Cross sectional area Moment of inertia Depth (diameter for circular cross section) End release code at node 1 End release code at node 2 Shear factor in the element y-axis Temperature difference in the element y-axis Perimeter (for thermal analysis only)
BEAM3D	14 to 27	RC1 RC2 … …	Cross sectional area Moment of inertia … … … Temperature difference in the element y-axis Perimeter (for thermal analysis only) … … … (Refer to the elements chapter in the COSMOS/M User Guide - Volume 1, for more information)
RBAR	1	RC1	Axial stiffness
SPRING	2	RC1 RC2	Axial stiffness Rotational stiffness
PIPE	3	RC1 RC2 RC3	Outer diameter Wall thickness Internal pressure
ELBOW	4	RC1 RC2 RC3 RC4	Outer diameter Wall thickness Internal pressure Radius of curvature
GAP	2	RC1 RC2	Allowable relative displacement between two nodes Coefficient of friction
MASS	7	RC1 RC2 RC3 RC4 RC5 RC6 RC7	Mass in x-direction Mass in y-direction Mass in z-direction Rotary inertia about x-axis Rotary inertia about y-axis Rotary inertia about z-axis Thermal capacity (in units of heat energy)

Table 6-11. Section Constants for All Elements in the Basic FEA System (Continued)

COSMOS/M Element Name	Section Property Constants Required		
	Number	RC	Description
BOUND	2	RC1 RC2	Axial stiffness Rotational stiffness
PLANE2D 4-NODE	2	RC1 RC2	Thickness (for plane stress problems only) Material angle
PLANE2D 8-NODE	2	RC1 RC2	Thickness (for plane stress problems only) Material angle
PLANE2D 4-NODE (Axisymmetric)	1	RC2	Material angle
PLANE2D 8-NODE (Axisymmetric)	1	RC2	Material angle
TRIANG 3-NODE	2	RC1 RC2	Thickness (for plane stress problems only) Material angle
TRIANG 6-NODE	2	RC1 RC2	Thickness (for plane stress problems only) Material angle
TRIANG 9-NODE	2	RC1 RC2	Thickness (for plane stress problems only) Material angle
TRIANG 3-NODE (Axisymmetric)	1	RC2	Material angle
TRIANG 6-NODE (Axisymmetric)	1	RC2	Material angle
TRIANG 9-NODE (Axisymmetric)	1	RC2	Material angle
SHELL3	3	RC1 RC2 RC3	Thickness Temperature gradient Foundation stiffness
SHELL4	3	RC1 RC2 RC3	Thickness Temperature gradient Foundation stiffness
SHELL3T	3	RC1 RC2 RC3	Thickness Temperature gradient Foundation stiffness
SHELL4T	3	RC1 RC2 RC3	Thickness Temperature gradient Foundation stiffness
SHELL3L	2+3NL	RC1 RCxx	*The number of real constants to be entered depends on the number of layers (NL). Refer to the elements chapter in the COSMOS/M User Guide - Volume 1, for more information.*

Table 6-11. Section Constants for All Elements in the Basic FEA System (Concluded)

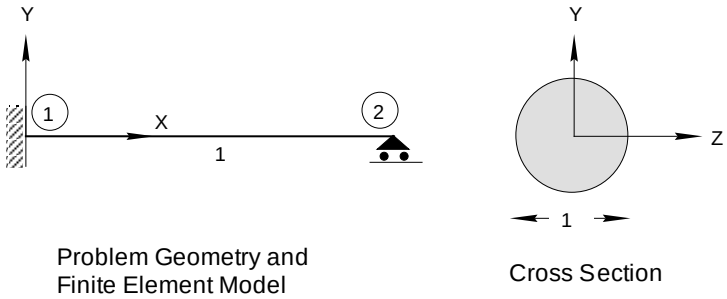
COSMOS/M Element Name	Section Property Constants Required		
	Number	RC	Description
SHELL4L	2+3NL	RC1 RCxx	Same as above
SHELL6	2	RC1 RC2	Thickness Temperature gradient
SHELL9	1	RC1	Thickness
SHELL9L	2+3NL	RC1 RCxx	*The number of real constants to be entered depends on the number of layers (NL). Refer to the elements chapter in the COSMOS/M User Guide - Volume 1, for more information.*
SHELLAX	1	RC1	Shell thickness
TETRA4	9	RC1-9	X,Y,Z coordinates of three points (for orthotropic materials only)
TETRA4R	9	RC1-9	X,Y,Z coordinates of three points (for orthotropic materials only)
TETRA10	9	RC1-9	X,Y,Z coordinates of three points (for orthotropic materials only)
SOLID	9	RC1-9	X,Y,Z coordinates of three points (for orthotropic materials only)
SOLIDL	2+6NL	RC1 RCxx	*The number of real constants to be entered depends on the number of layers (NL). Refer to the elements chapter in the COSMOS/M User Guide - Volume 1, for more information.*
SOLIDPZ	9	RC1-9	X,Y,Z coordinates of three points (for orthotropic materials only)
GENSTIF	78	RC1-78	The real constants are the numeric stiffness coefficients.

The Concept of Starting Locations for Real Constants

When you are entering the input for different section properties under the **RCONST** (Propsets > **Real Constant**) command, you will also be prompted for the starting location of the real constants. If you are defining the section properties for the first time, you need to start from the first location. However, if you make a mistake in entering the section properties, you do not need to start from the beginning for that real constant set. You can move to the specific location of the real constant that needs to be modified by using the starting

location option. This feature lets you modify or redefine selected values of the section properties without altering the remaining already input. For elements with many real constants, this feature is particularly helpful. Consider for example, a problem using BEAM2D element (which requires 8 real constants) as shown in the figure below:

Figure 6-7. Problem for Illustrating the Starting Locations of Real Constants



The section constants for the above BEAM2D element are defined as follows:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **8**

RC1 : Cross-sectional area > **0.785**
RC2 : Moment of inertia (Iz) > **0.0491**
RC3 : Depth [0] > **1**
RC4 : End-release code (node 1) > **0**
RC5 : End-release code (node 2) > **0**
RC6 : Shear factor in elem. y-axis > **0.9**
RC7 : Temp. diff. in elem. y-axis > **0**
RC8 : Perimeter (HSTAR only) > **3.1416**

The values input above will be stored in the proper locations of real constants as shown below:

RC1	RC2	RC3	RC4	RC5	RC6	RC7	RC8
0.785	0.0491	1	0	0	0	0	3.1416

Assume that you are now required to specify a temperature difference of 10 degrees between the top and bottom edges of the beam (along the element y-axis). As you can see from the above illustration for BEAM2D element, the temperature difference is stored in location RC7. Therefore you need to change *only* the real constant value stored under RC7 for this set. The procedure to accomplish this is illustrated below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set [2] > **1**

Start location of the real constants [9] > **7**
No. of real constants to be entered [2] > **1**

RC7 : Temp. diff. in elem. y-axis [0] > **10**

The above procedure will ensure that you are only altering the value stored under RC7, and the remaining real constants will be unaltered as shown below (shaded cells indicate constants defined earlier):

RC1	RC2	RC3	RC4	RC5	RC6	RC7	RC8
0.785	0.0491	1	0	0	0	10	3.1416

If you execute the **RCLIST** (Edit > LIST > **Real Constants**) command, the values shown above will be listed on the screen.

Explanation of Some Section Properties

For a detailed description of all real constants for any element type, refer to Chapter 4, *Element Library*, of COSMOS/M User Guide Volume 1. The following sections present further explanations on some of the section properties of elements in the Basic FEA System. In order to facilitate a clear presentation, the elements have been grouped as shown in the table below and their real constants are explained in the corresponding sections.

Element Name	**See Section…**
TRUSS2D, TRUSS3D, BEAM2D, BEAM3D, PIPE, & ELBOW	Section Properties for 1D Elements

PLANE2D, SHELL3, SHELL4, SHELL3T, SHELL4T, SHELL3L, SHELL4L, SHELL9, SHELL9L, SHELLAX, & TRIANG	Section Properties for 2D Continuum and Axisymmetric Elements
SOLID, SOLIDL, SOLIDPZ, TETRA4, TETRA4R & TETRA10	Section Properties for 3D Solid Elements
BOUND, GAP, GENSTIF, MASS, RBAR, & SPRING	Section Properties for Special Elements

Section Properties for 1D Elements

The section properties discussed under 1D element category include TRUSS2D, TRUSS3D, BEAM2D, BEAM3D, PIPE, and ELBOW elements.

There are only two section properties for TRUSS2D and TRUSS3D elements which include the cross sectional area (RC1) and the perimeter (RC2). However, as shown in Table 6-11, the section constants for beam elements can be quite complex, and they include end release codes, shear factors, and other input information which warrant further explanation. The following sections discuss some of the real constant information for beam elements to help you in better understanding of the element property input.

The shear factors (otherwise known as reduced area factors) for beam elements are used in computing the shear deflections and they are input as real constants for both BEAM2D and BEAM3D. These factors are 0.8333 for a solid rectangular section, 0.9 for a solid circular section, and 0.5 for a hollow circular section.

If you are using unsymmetric beam cross sections (supported for BEAM3D only), there are 27 real constants required. The node of a 3D beam element need not be located at the centroid of the cross section, and you can use real constants 15 through 27 for specifying the offset distances. Please refer to Chapter 4 of the COSMOS/M User Guide (V. 1).

The section constants for pipe elements in COSMOS/M (PIPE and ELBOW) include the outer diameter, wall thickness, and the radius of curvature. The internal pressure loading is treated as a section property so that for parts of the finite element model with different cross sectional dimensions, you can input

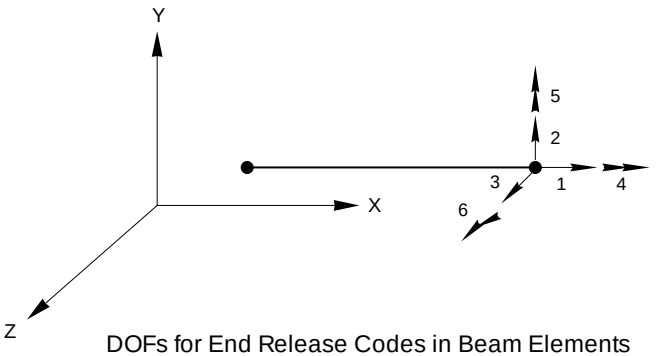



different values of pressure if necessary. Additionally, the inclusion of internal pressure as a real constant facilitates the application of external distributed loads on the curved pipe elements.

End-Release Codes for Beam Elements

For BEAM2D and BEAM3D elements, two of the real constant data includes the end release codes at nodes 1 and 2. These codes are provided to model internal hinges and rollers which are not necessarily located at supports. Support conditions can be modeled using the boundary conditions commands (see Chapter 7, *Loads and Boundary Conditions*).

Figure 6-8. Degrees of Freedom for End Release Codes



DOFs for End Release Codes in Beam Elements

As shown in the figure above, for a 3D beam element, there are six degrees of freedom at each node. The translational components (1 to 3, denoted by single arrows) occupy the first three locations of the end release code where as the rotational components (4 to 6, denoted by double arrows) occupy the last three locations. The end release codes are therefore six digit integers with combinations of *zeros* and *ones*. If you do not have any internal hinges or rollers, then the input for end release codes is zero (default). If a value of one is placed in a location, then the force or moment in that direction is assumed to be zero. For example, in order to model a roller along the z-axis, you need to input an end release code of 001. The trailing zeros can be omitted since they are the default values. Table 6-12 presents a summary of the different end release codes and the corresponding hinge/roller conditions represented by them.

Table 6-12. End Release Codes for Beam Elements

DOF	End Release Code	Description
ALL	000000	All components of forces are computed (default)
UX	100000	Force along the x-direction is zero (roller along x-axis)
UY	010000	Force along the y-direction is zero (roller along y-axis)
UZ	001000	Force along the z-direction is zero (roller along z-axis)
RX	000100	Moment along the x-direction is zero (hinge in y-z plane)
RY	000010	Moment along the y-direction is zero (hinge in x-z plane)
RZ	000001	Moment along the z-direction is zero (hinge in x-y plane)

Note that for the BEAM2D element, since there are only three degrees of freedom at each node (u_x, u_y and θ_z), the end release codes at locations 3, 4, and 5 are always assumed to be zero irrespective of your input.

Torsional and Maximum Shear Stress Constants for Beam Elements

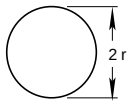
For BEAM3D element, one of the section constants input is the constant for maximum shear stress computations. This quantity designated as CTOR is input under RC14 for uniform symmetric sections, and under RC20 for symmetric tapered sections. The maximum shear stress is computed in terms of CTOR, torsion constant J, and the applied twisting moment T as shown below:

$$\tau_{max} = \frac{T \cdot TCOR}{J} \qquad \textbf{(6-29)}$$

Table 6-13 lists the constants J and CTOR for some standard cross sections.

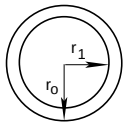
If you are using the command **BMSECDEF** (Propsets > **Beam Section**) to select standard sections for BEAM3D elements from the COSMOS/M section library, then CTOR is internally computed by the program.

Table 6-13. Torsional and Maximum Shear Stress Constants for Standard Cross Sections



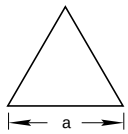
$J = \frac{\pi r^4}{2}$ CTOR = r

Solid Circular Section



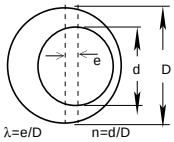
$J = \frac{\pi}{2}\left(r_o^4 - r_i^4\right)$ CTOR = 0.5 r_o

Hollow Concentric Circular Section



$J = \frac{a^4\sqrt{3}}{80}$ CTOR = 0.433 a

Solid Triangular Section (Equilateral)



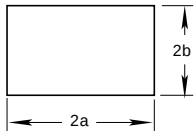
$J = \frac{\pi\left(D^4 - d^4\right)}{32\,Q}$ where

$Q = 1 + \frac{16n^2}{\left(1-n^2\right)\left(1-n^4\right)}\lambda^2 + \frac{384n^4}{\left(1-n^2\right)^2\left(1-n^4\right)^4}\lambda^4$

Hollow Eccentric Circular Section

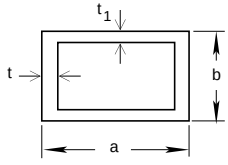
$J = 2.25\ a^4$

Solid Square Section



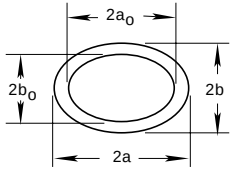
$J = ab^3\left[\frac{16}{3} - 3.36\frac{b}{a}\left(1 - \frac{b^4}{12\,a^4}\right)\right]$ for $a \geq b$

Solid Rectangular Section



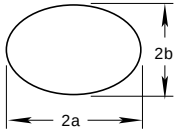
$J = \frac{2\,tt_1(a-t)^2(b-t_1)^2}{at + bt_1 - t^2 - t_1^2}$

Hollow Rectangular Section



$J = \frac{\pi a^3 b^3}{a^2 + b^2}\left(1 - q^4\right)$ where $q = \frac{a_o}{a} = \frac{b_o}{b}$

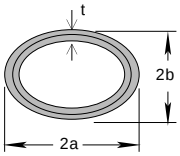
Hollow Elliptical Section



$J = \frac{\pi\, a^3\, b^3}{a^2 + b^2}$ CTOR = $\frac{2\,a^2\,b}{a^2 + b^2}$

Solid Elliptical Section

Table 6-13. Torsional and Maximum Shear Stress Constants for Standard Cross Sections (Concluded)

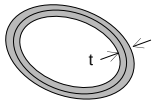


$$J = \frac{4\pi^2 t\left[\left(a - \frac{1}{2}t\right)^2 \left(b - \frac{1}{2}t\right)^2\right]}{U}$$

where **U** is the length of elliptical median boundary shown in dashed lines.

$$U = \pi(a + b - t)\left|1 + 0.258\frac{(a-b)^2}{(a+b-t)^2}\right|$$

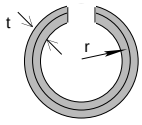
Hollow Thin Elliptical Section



$$J = \frac{4A^2 t}{U}$$

where **U** is length of median boundary and **A** is area within median boundary

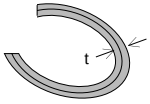
Any Thin Tube Section (Uniform Thickness)



$$J = \frac{2}{3}\pi rt^3$$

where r is the mean radius

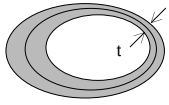
Thin Circular Open Tube Section



$$J = \frac{1}{3}Ut^3$$

where **U** is length of median line

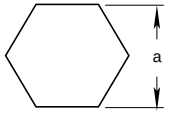
Any Thin Open Tube Section (Uniform Thickness)



$$J = \frac{4A}{\int dU/t}$$

where **A** is the area within median boundary and **t** is the thickness at any *point*

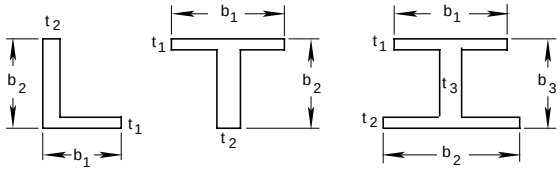
Any Thin Open Tube Section (Variable Thickness)



$$J = \frac{a^4}{8.8}$$

$$CTOR = 0.648\ a$$

Solid Hexagonal Section



$$J = \frac{1}{3}\sum_{i=1}^{n} b_i t_i^3 \quad CTOR = \mathrm{Max}\left(t_i,\ i=1,\ n\right)$$

Thin Walled Open Sections (Constant Thickness)

Section Properties for 2D Continuum and Axisymmetric Elements

The section properties discussed under 2D continuum and axisymmetric element category include PLANE2D, SHELL3, SHELL4, SHELL3T, SHELL4T, SHELL3L, SHELL4L, SHELL9, SHELL9L, SHELLAX, and TRIANG elements.

As seen in Table 6-11, for all 2D continuum elements (with the exception of composite shells), the first real constant (RC1) is the thickness. For PLANE2D and TRIANG (plane and axisymmetric) elements, the second real constant (RC2) is the material angle, required only if you are using the orthotropic material description. Whereas for all shell elements (except composites), the second real constant (RC2) represents the temperature gradient across the thickness. For SHELL3, SHELL4, SHELL3T and SHELL4T, the third real constant (RC3) represents the Foundation Stiffness. For a non-zero value for this real constant, the program attaches a spring at each node, applying a stiffness in the out-of plane direction. For more information on this constant refer to SHELL4 in Chapter 4, *Element Library*, of the COSMOS/M User Guide (Volume 1).

The procedure for defining real constants for most 2D continuum and axisymmetric element is quite simple. However, for laminated composite shell elements (SHELL3L, SHELL4L, and SHELL4L), the thickness of each layer, material group number, and the orientation of material principal directions are specified depending on the number of layers. Please refer to the section, *Material Properties for Laminated Composites*, for more information on defining real constants for each layer. That section also illustrates how you can rotate the orthotropic material axes with respect to the element local coordinate system or orient them with respect to the global Cartesian coordinate system.

✍ The thickness of each layer is uniform in composite shell elements. If you need to model variable thickness in each layer, you are recommended to use the composite solid element, SOLIDL.

Section Properties for 3D Solid Elements

The section properties discussed under 3D solid elements category include SOLID, SOLIDL, SOLIDPZ, TETRA4, TETRA4R, and TETRA10 elements.

If the material property is *isotropic*, then the sectional properties are *not* required to be input for all 3D solid elements. The *only* exception is the laminated composite solid element, SOLIDL. For this element, the thickness of each layer and the associated material property information (whether isotropic or orthotropic) are defined as real constants.

Consider an example using SOLIDL elements with *three* layers. It is assumed that all three layers have different material properties defined using the **MPROP** (Propsets > **Material Property**) command. Before proceeding to input the real constants, you need to first define the element group where you also specify the number of layers, as illustrated below:

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **SOLIDL: Composite 8-node solid element**

OP1 : Integr Type:1=Hybrid;2=Full > **Full**
OP2 : Number of Layers > **3**
OP3 : Stress Dir:0=Global;2=Mat > **Global**

Referring to Table 6-11, the number of real constants to be entered for a SOLIDL element with three layers is 20. The real constants for individual layers are defined as illustrated below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **10**

RC1 : Thickness flag (0:thickness,1:ratio) > **Thickness**
RC2 : Symmetric flag (0:all,1:sym,2:antisym) > **All**
RC3 : Material number for layer 1 > **1**
RC4 : Material angle for layer 1 > **0**
RC5 : Thickness (ratio) of layer 1 along 1-5 > **0.2**
RC6 : Thickness (ratio) of layer 1 along 2-6 > **0.25**
RC7 : Thickness (ratio) of layer 1 along 3-7 > **0.3**
RC8 : Thickness (ratio) of layer 1 along 4-8 > **0.35**
RC9 : Material number for layer 2 > **2**
RC10 : Material angle for layer 2 > **0**

You need to repeat the **RCONST** (Propsets > **Real Constant**) command, specify the same real constant set number, and input the remaining constants from starting

location 11. Since the SOLIDL element allows for the input of variable thickness in each layer, the input options are slightly different from those for composite shell elements.

If you are using solid elements with orthotropic material description, then the orientation of the orthotropic material axes need to be defined as real constants. The 9 real constants to be defined in this case the x-, y-, and z-coordinates of three points. Note that orthotropic material axes can be oriented *only* with respect to the default element coordinate system (ECS =-1). Please refer to Chapter 4, *Element Library*, of the COSMOS/M User Guide (Volume 1).

Section Properties for Special Elements

The section properties discussed under the special elements category include BOUND, GAP, GENSTIF, MASS, RBAR, and SPRING elements. These elements are termed special in the sense that for most frequently used finite element analyses, they are sparingly used.

The section properties discussed so far for one-, two-, and three-dimensional elements are mainly geometric in nature. However, as remarked earlier, the **RCONST** (Propsets > **Real Constant**) command is also used to define some physical properties of special elements in COSMOS/M. They include properties such as mass, stiffness coefficients, and frictional constants described in the following sections.

Mass Element

The mass element (MASS) in COSMOS/M represents a concentrated mass that can be defined at a node and it uses the lumped mass representation. However, unlike lumped masses which normally do not have rotary inertia, you can define rotary inertia in all three directions for the mass element. The real constants of a mass element therefore represent the diagonal terms of the lumped mass matrix. Additionally, a seventh real constant is provided to define thermal capacity for use in heat transfer analysis.




Before defining the real constants for a mass element, you need to first define the element (**EGROUP** (Propsets > **Element Group**)) as illustrated below:

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **MASS: General mass element**

OP1 : Unused option >

Next, use the **RCONST** (Propsets > **Real Constant**) command to define the components of mass and rotary inertia as illustrated below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants [1] > **1**
No. of real constants to be entered [7] > **7**

RC1 : Mass in x-direction > **1**
RC2 : Mass in y-direction > **1**
RC3 : Mass in z-direction > **1**
RC4 : Rotary inertia about x-axis > **2**
RC5 : Rotary inertia about y-axis > **2**
RC6 : Rotary inertia about z-axis > **2**
RC7 : Thermal capacity [0] >

Rigid Bar Element

The 2-node rigid bar element (RBAR) can be used to model infinitely rigid members in a finite element model. The element is used to transmit rigid body translations and rotations between two nodes. This element is defined as follows using the **EGROUP** (Propsets > **Element Group**) command.

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **RBAR: Rigid bar element**

OP1 : Unused option >

The real constant information for the rigid bar element consists of the bar stiffness, normally set to a high value as shown below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **1**

RC1 : Rigid bar stiffness [1e+10] > **2E15**

Boundary Element

The boundary element (BOUND) is a 2-node element (third node is required for the element orientation) with one of the nodes fixed while the other has two degrees of freedom, a translation and a rotation. The real constants for this element therefore consist of the axial and rotational stiffnesses. This element is defined as shown below using the **EGROUP** (Propsets > **Element Group**) command.

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **BOUND: Boundary Element**

OP1 : Bound. Type:0=Axial;1=Rot;2=Axial & Rot [0] > **2**

The corresponding real constants are defined as illustrated below using the **RCONST** (Propsets > **Real Constant**) command.

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **2**

RC1 : Axial stiffness > **1.5E6**
RC2 : Rotational stiffness > **2E5**

Spring Element

The spring element (SPRING) can be used to model translational and/or rotational actions in one-, two-, and three-dimensional applications. The real constants for this element consist of axial and torsional stiffnesses. This element is defined as shown below using the **EGROUP** (Propsets > **Element Group**) command.

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **SPRING: Spring element**

OP1 : Spring Type:0=Axial;1=Rot;2=Both > **2**
OP2 : Number of nodes:1=1-Node;2=2-Node > **2-Node**

The associated real constants for this element are defined as illustrated below using the **RCONST** (Propsets > **Real Constant**) command.

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **2**

RC1 : Axial stiffness > **1.5E6**
RC2 : Rotational stiffness > **2.E5**

You can notice from the above that the real constant input for the boundary element is the same as that of the spring element.

Gap Element

The gap element in the Basic FEA System (GAP) is a node-to-node interface element capable of modeling contact problems with or without friction and sliding. The gap element is defined as illustrated below using the **EGROUP** (Propsets > **Element Group**) command.

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **GAP: Gap element**

OP1 : Surface coord. update:0=No;1=Yes [0] > **No**
OP2 : Friction:0=No;1=Yes;2=Yes & Slide [0] > **Yes & Slide**
OP3 : Frict. outside X-Y plane:0=No;1=Yes [0] > **No**
OP4 : Gap Type:0=N-N;1=N-L;2=N-S [0] > **N-N**
OP5 : Unused option >
OP6 : Compressive gap measurement:0=User;1=Auto [0] > **User**

The last flag enables the user to avoid inputting the distance that the two nodes of each compressive gap have to move towards each other before resistance starts. Manual input is used in this example.

The real constants for the gap elements are defined as illustrated below:

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants [1] > **1**
No. of real constants to be entered [7] > **7**

RC1 : Relative displ. between 2 nodes [0] > **0.1**
RC2 : Coefficient of friction [0] > **0.5**
RC3 : Gap stiffness 0=No Spring k=Stiffness [0] > **1.5E4**
RC4 : Spring pre-load [0] > **1000**
RC5 : Maximum spring deflection [1e+08] > **10000000**
RC6 : Damping constant (c) [0] > **0**
RC7 : Damping constant (p) [1] > **1**

Refer to Chapter 4, *Element Library*, of the COSMOS/M User Guide (Volume 1) for more information on the gap element.

General Stiffness Element

The general stiffness element in COSMOS/M is a 2-node three dimensional element whose stiffness coefficients need to be supplied by the user for structural models. The real constants (a total of 78) for the GENSTIF element are the stiffness coefficients stored as shown in the matrix below:

k1,1(r1)	k1,2(r2)	k1,3(r3)	k1,4(r4)	k1,5(r5)	k1,6(r6)	k1,7(r7)	k1,8(r8)	k1,9(r9)	k1,10(r10)	k1,11(r11)	k1,12(r12)
	k2,2(r13)	k2,3(r14)	k2,4(r15)	k2,5(r16)	k2,6(r17)	k2,7(r18)	k2,8(r19)	k2,9(r20)	k2,10(r21)	k2,11(r22)	k2,12(r23)
		k3,3(r24)	k3,4(r25)	k3,5(r26)	k3,6(r27)	k3,7(r28)	k3,8(r29)	k3,9(r30)	k3,10(r31)	k3,11(r32)	k3,12(r33)
			k4,4(r34)	k4,5(r35)	k4,6(r36)	k4,7(r37)	k4,8(r38)	k4,9(r39)	k4,10(r40)	k4,11(r41)	k4,12(r42)
				k5,5(r43)	k5,6(r44)	k5,7(r45)	k5,8(r46)	k5,9(r47)	k5,10(r48)	k5,11(r49)	k5,12(r50)
					k6,6(r51)	k6,7(r52)	k6,8(r53)	k6,9(r54)	k6,10(r55)	k6,11(r56)	k6,12(r57)
						k7,7(r58)	k7,8(r59)	k7,9(r60)	k7,10(r61)	k7,11(r62)	k7,12(r63)
							k8,8(r64)	k8,9(r65)	k8,10(r66)	k8,11(r67)	k8,12(r68)
								k9,9(r69)	k9,10(r70)	k9,11(r71)	k9,12(r72)
									k10,10(r73)	k10,11(r74)	k10,12(r75)
										k11,11(r76)	k11,12(r77)
											k12,12(r78)

The above matrix also shows the corresponding real constant numbers (in parentheses) of the stiffness components. Before using the **RCONST** (Propsets > **Real Constant**) command, make sure that the GENSTIF element has been defined using the **EGROUP** (Propsets > **Element Group**) command. When entering the real constants, accept the number of real constants to be entered as

10, as illustrated below. The program will automatically remind you that you need 68 more constants to completely describe the GENSTIF element.

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start location of the real constants > **1**
No. of real constants to be entered > **10**

RC1 : K 1,1 stiffness term > **2.0E10**
RC2 : K 1,2 stiffness term > **2.5E8**
RC3 : K 1,3 stiffness term > **1.5E10**
RC4 : K 1,4 stiffness term > **2.8E4**
RC5 : K 1,5 stiffness term > **4E10**
RC6 : K 1,6 stiffness term > **3E12**
RC7 : K 1,7 stiffness term > **5E10**
RC8 : K 1,8 stiffness term > **6E8**
RC9 : K 1,9 stiffness term > **7E6**
RC10 : K 1,10 stiffness term > **6E10**
RC set 2 may need 68 more constants

To define the remaining constants, re-execute the **RCONST** (Propsets > **Real Constant**) command, specify the same set number and continue from starting location of 11 onwards.

Section Library

Similar to the material library feature, there are two different built-in section libraries in COSMOS/M for use with beam elements, BEAM2D and BEAM3D. These include the AISC (American Institute of Steel Construction, 9th Edition, 1991) sections for standard shapes, and the COSMOS/M sections for standard as well as non-standard (user-defined) shapes. The commands **BMSECDEF** (Propsets > **Beam Section**) and **PICK_SEC** (Propsets > **AISC Sect Table**) are used to select the sections and their properties from the COSMOS/M and AISC built-in section libraries respectively. When you use these commands, you do ***not*** need to use the **RCONST** (Propsets > **Real Constant**) command for defining the real constants. The following sections provide more information on the different cross sections and the procedure to use these sections.




✍ The COSMOS/M section library can be used with *both* BEAM2D and BEAM3D elements where as the AISC section library is applicable to BEAM3D element *only.*

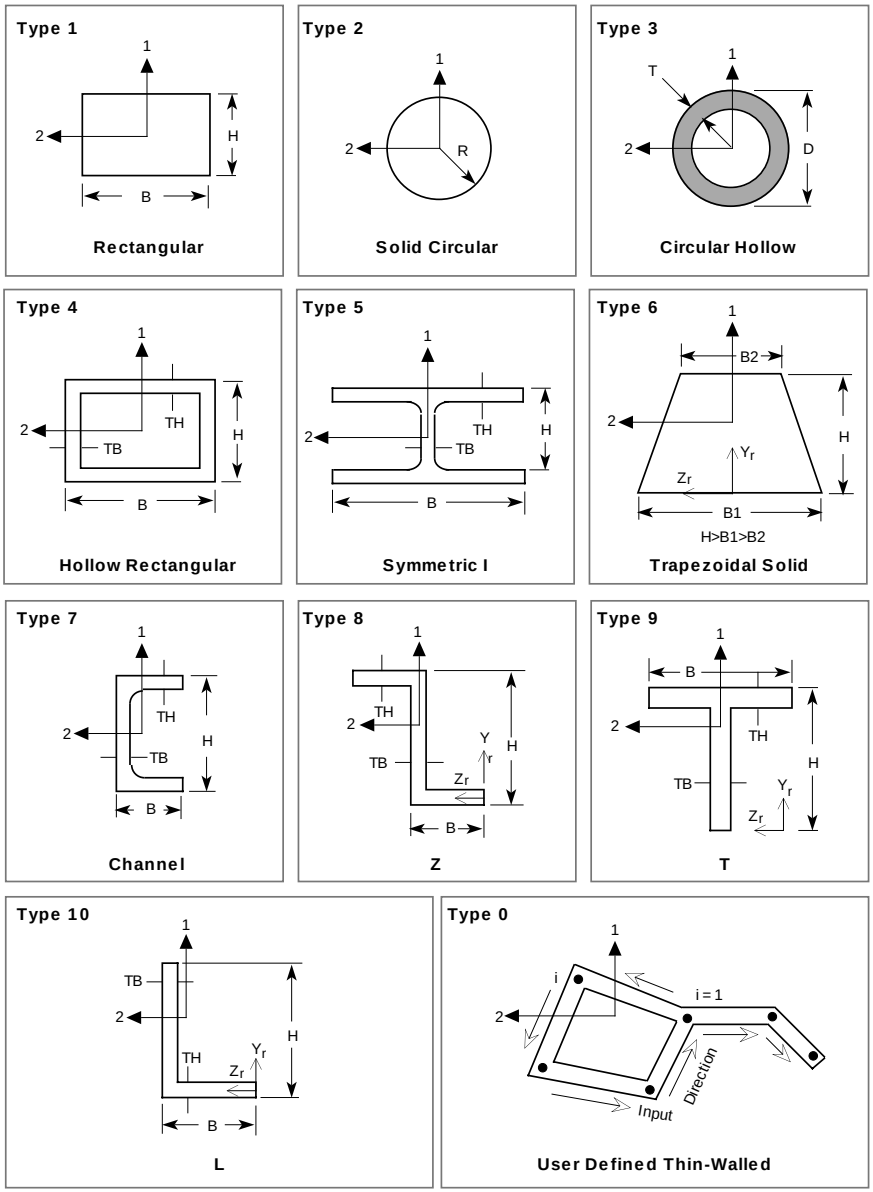
Using the Built-In COSMOS/M Section Library

In order to use the built-in COSMOS/M section library, you *must* first define the required beam element and choose the appropriate option, i.e., whether symmetric, unsymmetric, or tapered symmetric cross section. You can then use the **BMSECDEF** (Propsets > **Beam Section**) command illustrated below to select the section type and the cross sectional dimensions. For this command, you need to provide the associated element group number, the real constant set number, and the shape of the cross-section (described in the table below). The **BMSECDEF** (Propsets > **Beam Section**) command is used *in lieu* of **RCONST** (Propsets > **Real Constant**), and as such it also defines the end release codes and other real constant data for beam elements. Table 6-15 in the following page shows the shapes of cross sections and their identification parameters.

Table 6-14. Available Cross Sections in the COSMOS/M Section Library

Section Number	Description	Elements Supported	Remarks
1	Solid rectangular section	BEAM2D, BEAM3D	All symmetry options are valid for BEAM3D
2	Solid circular section	BEAM2D, BEAM3D	All symmetry options are valid for BEAM3D
3	Hollow circular section	BEAM2D, BEAM3D	All symmetry options are valid for BEAM3D
4	Hollow rectangular section	BEAM2D, BEAM3D	All symmetry options are valid for BEAM3D
5	Symmetric I-section	BEAM2D, BEAM3D	All symmetry options are valid for BEAM3D
6	Solid trapezoidal section	BEAM3D	BEAM3D must be unsymmetric
7	Channel section	BEAM3D	BEAM3D must be unsymmetric
8	Z-section	BEAM3D	BEAM3D must be unsymmetric
9	T-section	BEAM3D	BEAM3D must be unsymmetric
10	L-section	BEAM3D	BEAM3D must be unsymmetric
0	User-defined section	BEAM3D	BEAM3D must be unsymmetric

Table 6-15. Identification Parameters of the Built-In COSMOS/M Section Library



As seen in Table 6-14, the first five sections are symmetric and are applicable to both BEAM2D and BEAM3D elements. Sections 6 through 10 and 0 are applicable to BEAM3D elements *only*. Further, some of these sections have restrictions placed on the proportions of cross sectional dimensions. The trapezoidal section for example (section number 6) has a restriction that the depth should be greater than the width at bottom which in turn should be greater than the top width. You need to keep this restriction in mind when using a trapezoidal section.

When you use the **BMSECDEF** (Propsets > **Beam Section**) command for BEAM3D elements, the torsional and maximum shear stress constants shown in Table 6-13 are automatically computed by the program and your input is not required. Since the beam sections are associated with a real constant set, you need to use the **RCDEL** (Edit > DELETE > **Real Constants**) command to delete a beam section selected.

The following illustration demonstrates the selection of a hollow circular section (number 3 from Table 6-14) for the BEAM2D element. In order to use this section, you must define the element group: Next, the **BMSECDEF** (Propsets > **Beam Section**) command is used to select this cross section as illustrated below.

Geo Panel: Propsets > **Beam Section**
Associated Element group > **1**
Real Constant set > **1**

Section number [1] > **3**

Start location of the section parameters > **1**

No. of beam section constants to be entered > **7**

BS1 : Outside diameter (D) > **1**
BS2 : Thickness (T) > **0.1**
BS3 : End-release code (node 1) > **0**
BS4 : End-release code (node 2) > **0**
BS5 : Shear factor in elem. y-axis > **0**
BS6 : Temp. diff. in elem. y-axis > **0**
BS7 : Perimeter (HSTAR only) > **0**

Note that if you need to model any internal hinge or roller, you need to specify the appropriate input under the **BMSECDEF** (Propsets > **Beam Section**) command itself. The command **BMSECLIST** (Edit > LIST > **Beam Section**) can be used to list the section you selected as well as other real constant data input. The listing obtained on the screen will be of the form:

```
                          Real Constant Sets

Real Constant Set :   1   (ACTIVE)
Associated Element Group :   1   ( BEAM2D  -  Symmetric  Pipe )
Center of Gravity : ( cgy = 0.000000e+00    cgz = 0.000000e+00 )

  Bs1 : Outside diameter (D) . . . . . . . . . =   1.000000e+00
  Bs2 : Thickness  (T) . . . . . . . . . . . . =   1.000000e-01
  Bs3 : End-release code (node 1)  . . . . . . =              0
  Bs4 : End-release code (node 2)  . . . . . . =              0
  Bs5 : Shear factor in elem. y-axis  . . . . . =   0.000000e+00
  Bs6 : Temp. diff. in elem. y-axis   . . . . . =   0.000000e+00
  Bs7 : Perimeter (HSTAR only)  . . . . . . . . =   0.000000e+00
```

To see the computed cross sectional data such as area and moment of inertia, you need to use **RCLIST** (Edit > LIST > **Real Constants**) command as before. The cross sectional properties computed will be listed along with other real constants for the element as shown below:

```
                          Real Constant Sets

Real Constant Set :   1   (ACTIVE)
Associated Element Group :   1   ( BEAM2D )

  Rc1 : Cross-sectional area . . . . . . . . . =   2.827433e-01
  Rc2 : Moment of inertia (Iz)  . . . . . . . . =   2.898119e-02
  Rc3 : Depth . . . . . . . . . . . . . . . . . =   1.000000e+00
  Rc4 : End-release code (node 1)  . . . . . . =              0
  Rc5 : End-release code (node 2)  . . . . . . =              0
  Rc6 : Shear factor in elem. y-axis  . . . . . =   0.000000e+00
  Rc7 : Temp. diff. in elem. y-axis   . . . . . =   0.000000e+00
  Rc8 : Perimeter (HSTAR only)  . . . . . . . . =   0.000000e+00
```

The procedure for using symmetric tapered and unsymmetric beam sections (6 through 10 and 0) is similar. Since these types of sections are applicable to BEAM3D elements only, you need to define this element first and specify the appropriate section type, before using the **BMSECDEF** (Propsets > **Beam Section**) command.

Using the Built-In AISC Section Library

If you are analyzing steel framed structures with standard AISC (American Institute of Steel Construction) cross sections, COSMOS/M provides you with the capability to select various properties such as center of gravity, area, moment of inertia, and others from a built-in section library. The **PICK_SEC** (Propsets > **AISC Sect Table**) command assigns a specified section from the AISC section tables to a real constant set. *This command should only be used in conjunction with the FPS system*. The unit of length is inches and that of weight is lb/ft. The "*section name*" option in this command refers to the valid AISC sections shown in the table below.

Table 6-16. Available AISC Sections in the Built-In Section Library

Section Name	Description	Example
ANGLE	Standard angle sections	L 9x 4 x 5/8
CBEAM	American standard channel sections	C 15 x 50
HPBEAM	I-sections with equal flange and web thicknesses	HP 14 x 117
MBEAM	Miscellaneous I-sections	M 14 x 18
MCBEAM	Miscellaneous channel sections	MC 18 x 58
PIPE	Hollow circular pipe with uniform thickness	12 x 0.375
RANGLE	Angle sections with rotated properties	L 9x 4 x 5/8
SBEAM	I-sections with narrower flanges	S 24 x 121
STUBE	Hollow rectangular tube with uniform thickness	16 x 16
WBEAM	I-section with different flange and web thicknesses	W 36 x 300

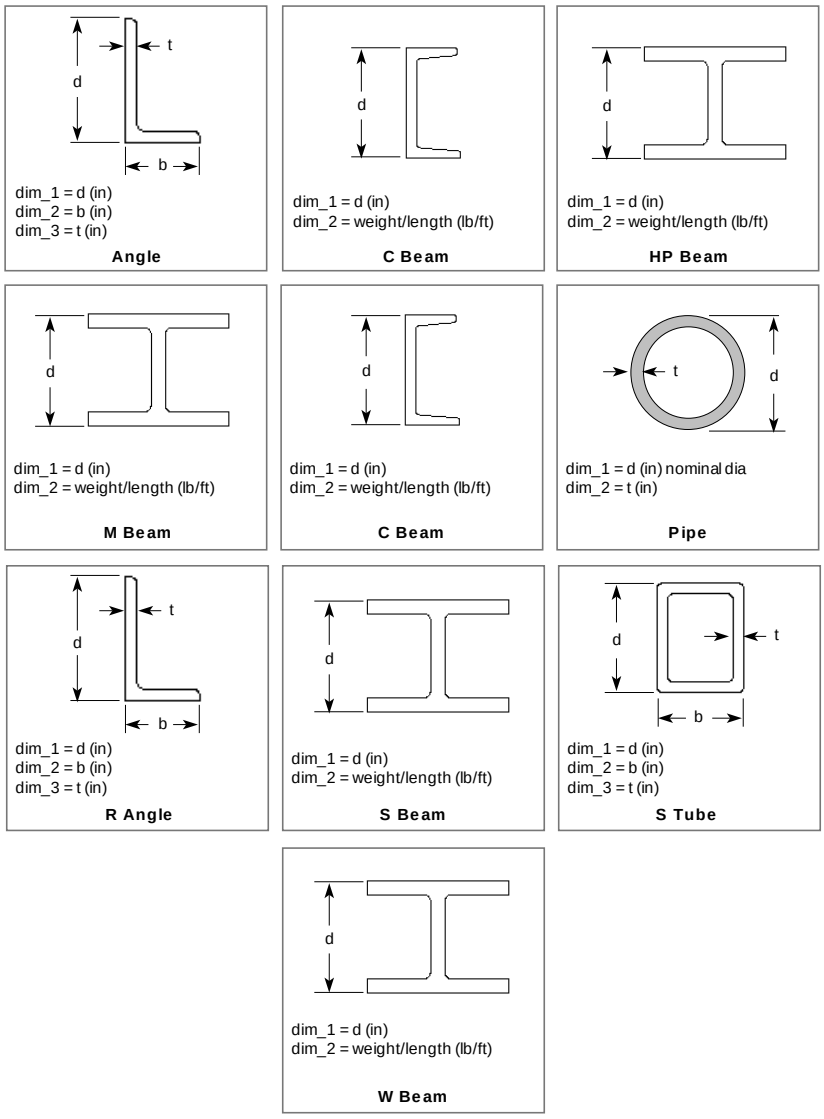
Also, in the above command, dim_1, dim_2, and dim_3 are the identification parameters for the section according to AISC. For example, if you want to select a W-section 36x300, then dim_1 is 36, and dim_2 is 300. The argument dim_3 is required only for ANGLE, RANGLE, and STUBE sections. Note that the definitions of local x, y, and z coordinates are different in GEOSTAR and the AISC code. Table 6-17 shown on the next page, graphically illustrates the identification parameters dim_1, dim_2, and dim_3 for the available sections.

The weight of the section for sections other than angles is based on the density of steel. If you are using some other material in your finite element model, then the weight should be based on steel as in the AISC code. This does *not* affect any material properties since the weight is used only to define the section properties.

✍ The AISC sections can be currently used for BEAM3D elements *only*.

It is possible to check if the structure you model and analyze using COSMOS/M meets the AISC specifications for allowable stress design (ASD). This is explained in the next section.

Table 6-17. Identification Parameters of AISC Sections



To choose a standard AISC section in COSMOS/M is extremely simple: use the command **PICK_SEC** (Propsets > **AISC Sect Table**) and specify the section name (given earlier in Table 6-16), as illustrated below:

Geo Panel: Propsets > **AISC Sect Table**
Associated Element group > **1**
Real Constant set > **1**
Section Name > **WBEAM**
AISC Dimension 1 > **36**
AISC Dimension 2 > **300**

You can use the command **RCLIST** (Edit > LIST > **Real Constants**) to verify the section properties accessed from the built-in AISC section library. The listing on the computer screen will be as shown below:

```
                        Real Constant Sets

 Real Constant Set :   1   (ACTIVE)
 Associated Element Group :   1   ( BEAM3D )

   Rc1 : Cross-sectional area . . . . . . . . =   8.830000e+01
   Rc2 : Moment of inertia about y-axis (Iy) . . . =   1.300000e+03
   Rc3 : Moment of inertia about z-axis (Iz) . . . =   2.030000e+04
   Rc4 : Depth of beam (y-axis)  . . . . . . . =   3.674000e+01
   Rc5 : Width of beam (z-axis)  . . . . . . . =   1.665500e+01
   Rc6 : End-release code (node 1)  . . . . . . =              0
   Rc7 : End-release code (node 2)  . . . . . . =              0
   Rc8 : Moment of inertia about x-axis (Ix) . . . =   6.420000e+01
```

AISC Code Check (Using Steel Design Module)

If AISC standard sections are used in the analysis, you can specify an option in the **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) command to verify the validity of the chosen sections according to AISC provisions; see illustration below:

Geo Panel: Analysis > STATIC > **Stress Analysis Options**
AISC code check Flag 0=No 1=Yes >**Yes**
... ... >...
Accept Defaults

When the AISC code check flag is turned on, the program generates a file, jobname.AIS. This file can be used with the code checking program SDM (Steel Design Module) which is a *stand alone* program for applying the AISC code criteria (contact Structural Research for availability).

Modification, Display and Output of Section Property Definitions

The command **RCDEL** (Edit > DELETE > **Real Constants)** can be used to delete *all* properties defined in one or more real constant sets. If you like to delete or modify a specific property in a real constant set, then you need to make use of the starting locations concept explained earlier.

If you have more than one real constant set in your model and would like to make a particular set active, then you need to use the command **ACTSET** (Control > ACTIVATE > **Set Entity**). In addition to activating real constant sets, this command can be used to activate sets for coordinate systems, temperature curves, and many others. The description of these sets and their set names is shown in the Table 6-10. The set names that are currently active can be graphically viewed on the screen using the **STATUS1** icon from Control > UTILITY, as shown in Figure 6-4.

To change the association of *existing* real constant sets with element groups and material property sets, the **EPROPCHANGE** (Propsets > **Change El-Prop**) command can be used. This command also assigns a specified color to a selected pattern of elements. The assigned colors can be subsequently used to display and hence identify parts of your mesh with different real constant sets. The next section explains this in more detail.

Display of Section Property Definitions

For models with more than one real constant sets, COSMOS/M provides a novel way to graphically represent the different section constant sets for model checking purposes. The command **ACTECLR** (Meshing > ELEMENTS > **Activate Elem Color**) can be used to activate the *default* colors for elements and one of the associated sets (the real constant set in this case). *The procedure for displaying different sectional properties is similar to that for material properties; please refer to that section for more information*.




Section Property Output

By default, the section property information is *not* written to the output file. However, you can request for a detailed output using the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command and specify the input print flag. The section properties defined can also be listed on the screen using the **RCLIST** (Edit > LIST > **Real Constants**) command. As with all listing commands in GEOSTAR, the information from **RCLIST** (Edit > LIST > **Real Constants**) command can be *routed* to an ascii file using the command **LISTLOG** (Control > MISCELLANEOUS > **List Log**).

Computing Mass and Inertia Properties

The **MASSPROP** (Control > MEASURE > **Find Mass Property**) command computes and lists mass and inertia properties for a pattern of elements. The properties listed are: length, area, volume, mass, and the corresponding centers of gravity, moments of inertia and radii of gyration. Length properties are listed for one dimensional elements *only* (e.g. BEAM3D, ELBOW, PIPE, etc.), area properties are listed for two dimensional elements *only* (e.g. SHELL9L, SHELLAX, FLOW2D, MAG2D, etc.), and volume properties are listed for all elements *with* volume (e.g. SOLID, FLOW3D, BEAM2D, SHELL4, etc.). Mass properties are listed for elements with mass (density must be specified). The command also computes the principal moments of inertia and the corresponding principal directions with respect to an arbitrary Cartesian coordinate system. The “check flag” option in this command if for checking inconsistent data such as undefined material properties, section constants, connectivity, and degenerate elements. You can choose to compute mass and inertia properties for all or some of the elements.

The algorithm for computing the mass and inertia properties uses definite integrals in one, two and three dimensions. These integrals are numerically evaluated for each element and summed over the total number of elements specified in the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command. The following sections provide more information on the computation of mass and inertia properties for lattice (framed), planar (flat and curved), and solid structures modeled using finite elements.




Computing the mass and inertia properties using **MASSPROP** (Control > MEASURE > **Find Mass Property**) command is quite simple. Simply issue this command *after* you have specified the element group, section constants, and material properties for your finite element mesh. It is a good practice to turn on the check flag to make sure the specified elements have all required properties defined, as illustrated below:

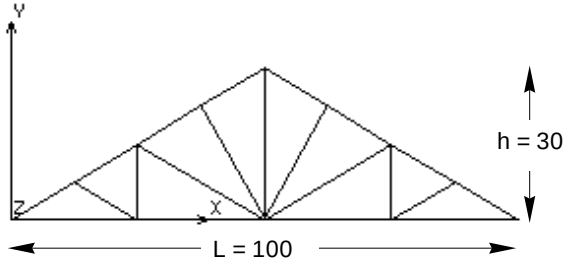
Geo Panel: Control > MEASURE > **Find Mass Property**
Check flag: 1: Yes; 0: No [0] > **1**
Beginning Element > **1**
Ending Element > **40**
Increment > **1**
Coordinate system > **0**

Some of the mass and inertia properties discussed in this section are also computed and printed in the output file when you perform natural frequency analysis or static analysis with gravity loading.

Mass and Inertia Property Computation of Lattice Structures

Consider the example of a truss shown in the figure below for computing the mass and inertia properties. The dimensions and mass density of the truss are also indicated in the figure. It is assumed that all elements are considered for mass property computations using **MASSPROP** (Control > MEASURE > **Find Mass Property**). The coordinate system indicated is used for computing the mass and inertia properties.

Figure 6-9. Example for Mass and Inertia Property Computations of a Lattice Structure



c/s area = 1
moment of inertia = 0.08333
mass density = 7.246E-04

The length, volume, and mass of lattice structures are computed using:

$$L=\sum_{i=1}^{nel} l_i, \qquad V=\sum_{i=1}^{nel} A_i l_i, \qquad M=\sum_{i=1}^{nel} \rho_i A_i l_i \tag{6-30}$$

where A_i and l_i are the cross sectional area and length of an element *i*, and *nel* is the total number of elements specified. The coordinates of the centroid are established by first computing the static moments and then dividing them by the total length, as shown below:

$$M_x=\sum_{i=1}^{nel} l_i y_{ci}\,, \qquad M_y=\sum_{i=1}^{nel} l_i x_{ci}\,, \qquad x_c=\frac{M_y}{L}, \qquad y_c=\frac{M_x}{L} \tag{6-31}$$

where x_{ci} and y_{ci} are the centroidal coordinates of an element. The information on length, volume, and mass will be listed as shown below when the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command is executed.

```
                Mass   Distribution   Table
 Physical Properties:  Centroids w.r.t. Coord. Sys. [0]:
                                        x         y         z
   Length = 415.561     CGL =  ( 50    , 8.54282    , 0 )
   Area   = 0           CGA =  ( 0     , 0          , 0 )
   Volume = 415.561     CGV =  ( 50    , 8.54282    , 0 )
   Mass   = 0.301116    CG  =  ( 50    , 8.54282    , 0 )
```

The equations for moments of inertia with respect to length are shown below:

$$I_{xx}=\sum_{i=1}^{nel}\int_{l_i}\left(y^2+z^2\right)\,dl_i+l_i\left(y_{ci}^2+z_{ci}^2\right), \quad I_{yy}=\sum_{i=1}^{nel}\int_{l_i}\left(x^2+z^2\right)\,dl_i+l_i\left(x_{ci}^2+z_{ci}^2\right) \tag{6-32}$$

The products of inertia for lattice models are computed using the equations shown below:

$$I_{xy}=I_{yx}=\sum_{i=1}^{nel}\int_{l_i} xy\;dl_i+l_i x_{ci} y_{ci} \tag{6-33}$$

The radius of gyration is computed as follows:

$$r_x=\sqrt{\frac{I_{xx}}{A}}, \qquad r_y=\sqrt{\frac{I_{yy}}{A}}, \qquad r_z=\sqrt{\frac{I_{zz}}{A}} \tag{6-34}$$

The moment and product of inertia with respect to the *specified* coordinate system and the corresponding radii of gyration will be listed as shown below when the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command is executed:

Moments of Inertia and Radii of Gyration (w.r.t. Coord. Sys. [0])

	Length	Area	Volume	Mass
Ixx	59842.5	0	59899	43.4028
Iyy	1.28185e+06	0	1.2819e+06	928.862
Izz	1.34169e+06	0	1.34173e+06	972.215
Ixy	177503	0	177503	128.619
Iyz	0	0	0	0
Izx	0	0	0	0
Rx	12.0002	0	12.0058	12.0058
Ry	55.5394	0	55.5404	55.5404
Rz	56.821	0	56.8217	56.8217

The moment and product of inertia are also listed with respect to the centroid in the *specified* coordinate system as shown below:

Moments of Inertia and Radii of Gyration (w.r.t. CG's in Coord. Sys.[0])

	Length	Area	Volume	Mass
Ixx	29514.9	0	29571.4	21.4274
Iyy	242946	0	242993	176.073
Izz	272461	0	272495	197.45
Ixy	-0.0235475	0	-0.0235475	-7.92031e-06
Iyz	0	0	0	0
Izx	0	0	0	0
Rx	8.42759	0	8.43565	8.43565
Ry	24.1789	0	24.1813	24.1813
Rz	25.6056	0	25.6072	25.6072

The principal moments of inertia are the maximum and minimum moments of inertia which can be measured about any axis which passes through a given point. The product of inertia is zero for any two axes, one of which is a principal axis. An axis of symmetry is *always* a principal axis. The listing below shows the principal moments of inertia with respect to the centroid.

Principal Moments of Inertia (w.r.t. CG's)

	Length	Area	Volume	Mass
I1	272461	0	272495	197.45
I2	242946	0	242993	176.073
I3	29514.9	0	29571.4	21.4274

The corresponding direction cosine vectors of the principal axes with respect to the *specified* coordinate system are listed below:

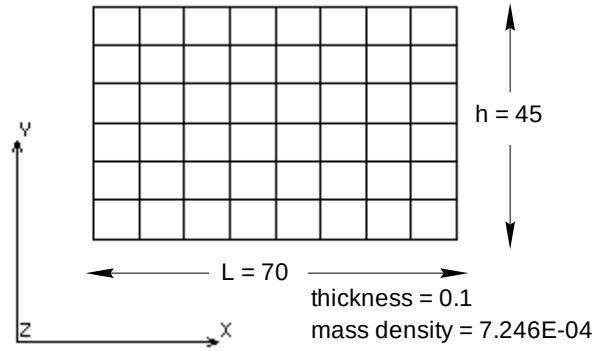
Direction Cosine Vectors (w.r.t. Coord. System [0])

	Length	Area	Volume	Mass
a11	0	0	0	0
a21	0	0	0	0
a31	1	0	1	1
a12	1.10329e-07	0	1.10333e-07	5.1216e-08
a22	1	0	1	1
a32	0	0	0	0
a13	-1	0	-1	-1
a23	1.10329e-07	0	1.10333e-07	5.1216e-08
a33	0	0	0	0

Mass and Inertia Property Computation of Plane Continuum Structures

Consider the example of a flat plate shown in the figure below to understand the mass property computations of two dimensional continuous (flat and curved) models. The dimensions of the plate and the mass properties are also shown. The bottom left corner of the plate is at coordinates (10,15) with respect to the coordinate system shown. The example chosen is quite simple so that you can verify the results obtained using **MASSPROP** (Control > MEASURE > **Find Mass Property**) by hand calculations. All elements shown in the figure will be considered in the coordinate system shown for mass property calculation.

Figure 6-10. Example for Mass and Inertia Property Computations of a Plane Structure



The area, volume, and mass of a planar structure are computed as follows:

$$A=\sum_{i=1}^{nel} A_i, \quad V=\sum_{i=1}^{nel} t_i\ A_i, \quad M=\sum_{i=1}^{nel} t_i\ \rho_i A_i \qquad \textbf{(6-35)}$$

where A_i is the area, t_i is the thickness and ρ_i the mass density of element *i*, and *nel* is the total number of elements specified. The coordinates of the centroid are established by first computing the static moments and then dividing them by the total area, as shown below:

$$M_x=\sum_{i=1}^{nel} A_i y_{ci}\ , \quad M_y=\sum_{i=1}^{nel} A_i x_{ci}\ , \quad x_c=\frac{M_y}{A}, \quad y_c=\frac{M_x}{A} \qquad \textbf{(6-36)}$$

The listing below shows the area, volume, mass and centroidal locations of the plate obtained using **MASSPROP** (Control > MEASURE > **Find Mass Property**) command:

```
                    Mass  Distribution  Table
Physical Properties:   Centroids w.r.t. Coord. Sys. [0]:
                                    x         y             z
Length = 0             CGL =   ( 0       , 0           , 0 )
Area   = 3150          CGA =   ( 50      , 42.5        , 0 )
Volume = 315           CGV =   ( 50      , 42.5        , 0 )
Mass   = 0.228249      CG  =   ( 50      , 42.4999     , 0 )
```

The moments and products of inertia for planar models are computed with respect to area, volume as well as mass. The equations for moment of inertia of a plane area are shown below:

$$I_{xx}=\sum_{i=1}^{nel}\iint_{A_i} y^2 dA_i + A_i\left(y_{ci}^2+z_{ci}^2\right), \quad I_{yy}=\sum_{i=1}^{nel}\iint_{A_i} x^2 dA_i + A_i\left(x_{ci}^2+z_{ci}^2\right) \qquad \textbf{(6-37)}$$

The products of inertia of a plane area are computed as shown below:

$$I_{xy}=I_{yx}=\sum_{i=1}^{nel}\iint_{A_i} xy\, dA_i + A_i x_{ci} y_{ci} \qquad \textbf{(6-38)}$$

The radius of gyration is computed using the equation 6-34 shown earlier. The moment and product of inertia with respect to the *specified* coordinate system and the corresponding radii gyration will be listed as shown below when the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command is executed:

Moments of Inertia and Radii of Gyration (w.r.t. Coord. Sys. [0])

	Length	Area	Volume	Mass
Ixx	0	6.22125e+06	622125	450.792
Iyy	0	9.16125e+06	916125	663.824
Izz	0	1.53825e+07	1.53825e+06	1114.62
Ixy	0	6.69375e+06	669375	485.029
Iyz	0	0	0	0
Izx	0	0	0	0
Rx	0	44.441	44.441	44.4409
Ry	0	53.929	53.929	53.9289
Rz	0	69.8808	69.8809	69.8808

The moment and product of inertia are also listed with respect to the centroid in the *specified* coordinate system as shown below:

Moments of Inertia and Radii of Gyration (w.r.t. CG's in Coord. Sys. [0])

	Length	Area	Volume	Mass
Ixx	0	531564	53156.4	38.5176
Iyy	0	1.28625e+06	128625	93.2023
Izz	0	1.81781e+06	181781	131.72
Ixy	0	-0.510693	-0.0510693	0.000538818
Iyz	0	0	0	0
Izx	0	0	0	0
Rx	0	12.9904	12.9904	12.9905
Ry	0	20.2072	20.2073	20.2073
Rz	0	24.0225	24.0225	24.0226

For plane objects, there exists two mutually perpendicular axes through any given origin about which the product of inertia are zero. These axes are called the principal axes. If ωrepresents the angle of rotation from the original coordinate axis, then the value of this angle to satisfy the condition that the product of inertia is zero is given by:

$$\tan 2\omega = -\frac{2 I_{xy}}{I_{xx} - I_{yy}} \qquad \textbf{(6-39)}$$

The principal moments are given by:

$$I_{1,2} = \frac{I_{xx} + I_{yy}}{2} \pm \frac{1}{2}\sqrt{\left(I_{xx} - I_{yy}\right)^2 + 4 I_{xy}^2} \qquad \textbf{(6-40)}$$

The principal moments of inertia are the maximum and minimum moments of inertia which can be measured about any axis which passes through a given point. The product of inertia is zero for any two axes, one of which is a principal axis. An axis of symmetry is *always* a principal axis. The listing below shows the principal moments of inertia with respect to the centroid.

Principal Moments of Inertia (w.r.t. CG's):

	Length	Area	Volume	Mass
I1	0	1.81781e+06	181781	131.72
I2	0	1.28625e+06	128625	93.2023
I3	0	531564	53156.4	38.5176




The corresponding direction cosine vectors of the principal axes with respect to the *specified* coordinate system are listed below:

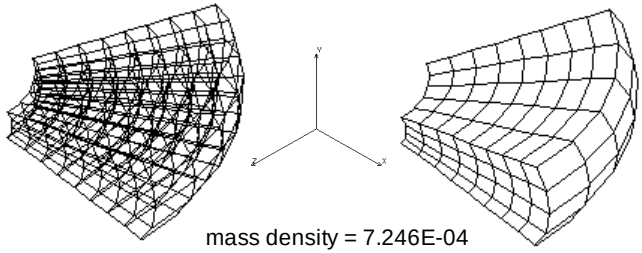
Direction Cosine Vectors (w.r.t. Coord. System [0])

	Length	Area	Volume	Mass
a11	0	0	0	0
a21	0	0	0	0
a31	0	1	1	1
a12	0	6.76698e-07	6.76696e-07	-9.85317e-06
a22	0	1	1	1
a32	0	0	0	0
a13	0	-1	-1	-1
a23	0	6.76698e-07	6.76696e-07	-9.85317e-06
a33	0	0	0	0

Mass and Inertia Property Computation of Solid Structures

Consider the following example of a solid finite element model with properties indicated. The model shown was built by first generating the finite element mesh of the plane cross section and sweeping this mesh about the Y-axis by an angle of 45 degrees to obtain a solid mesh of hexahedral elements. The command **PHEXTR** (Geometry > POLYHEDRA > **Extrusion**) was used for sweeping.

Figure 6-11. Example for Mass and Inertia Property Computations |of a Solid Structure



For solid models, volume and mass are computed as follows:

$$V = \sum_{i=1}^{nel} V_i , \quad M = \sum_{i=1}^{nel} \rho_i V_i \qquad \textbf{(6-41)}$$

where V_i is the volume, ρ_i is the mass density of element *i*, and *nel* is the total number of elements specified. To compute the centroidal information, the static moments are first computed as follows:

$$M_{xy} = \sum_{i=1}^{nel} \rho_i V_i Z_{ci}, \quad M_{yz} = \sum_{i=1}^{nel} \rho_i V_i X_{ci}, \quad M_{zx} = \sum_{i=1}^{nel} \rho_i V_i y_{ci} \tag{6-42}$$

Coordinates of the centroid are then computed using:

$$x_c = \frac{M_{yz}}{V}, \quad y_c = \frac{M_{zx}}{V}, \quad z_c = \frac{M_{xy}}{V} \tag{6-43}$$

The computed mass, volume, and centroidal information will be listed as shown below when you use the **MASSPROP** (Control > MEASURE > **Find Mass Property**) command.

```
Mass   Distribution   Table
 Physical Properties:          Centroids w.r.t. Coord. Sys. [0]
                                    x          y          z
  Length = 0             CGL =   ( 0         , 0         , 0        )
  Area   = 0             CGA =   ( 0         , 0         , 0        )
  Volume = 92140.4       CGV =   ( 59.5725   , 25        , -34.3942)
  Mass   = 66.7649       CG  =   ( 59.5725   , 24.9999   , -34.3942)
```

The mass moments of inertia are computed as follows:

$$I_{xx} = \sum_{i=1}^{nel} \rho_i \left[\iiint_{V_i} (y^2+z^2) dV_i + V_i (y_{ci}^2 + z_{ci}^2) \right],$$

$$I_{yy} = \sum_{i=1}^{nel} \rho_i \left[\iiint_{V_i} (x^2+z^2) dV_i + V_i (x_{ci}^2 + z_{ci}^2) \right], \tag{6-44}$$

$$I_{zz} = \sum_{i=1}^{nel} \rho_i \left[\iiint_{V_i} (x^2+y^2) dV_i + V_i (x_{ci}^2 + y_{ci}^2) \right]$$

The radii of gyration are computed using equation 6-34 shown earlier. The products of inertia are computed as follows:

$$I_{yz}=I_{zy}\sum_{i=1}^{nel}\rho_i\left[\iiint_{V_i} yz\,dV_i + V_i\,y_{ci}\,z_{ci}\right],$$

$$I_{xz}=I_{zx}\sum_{i=1}^{nel}\rho_i\left[\iiint_{V_i} xz\,dV_i + V_i\,x_{ci}\,z_{ci}\right], \qquad \textbf{(6-45)}$$

$$I_{xy}=I_{yx}\sum_{i=1}^{nel}\rho_i\left[\iiint_{V_i} xy\,dV_i + V_i\,x_{ci}\,y_{ci}\right]$$

The computed moment and product of inertia along with radii of gyration with respect to the *specified* coordinate system will be listed as shown below:

Moments of Inertia and Radii of Gyration (w.r.t. Coord. Sys. [0])

	Length	Area	Volume	Mass
Ixx	0	0	2.08959e+08	151412
Iyy	0	0	5.01588e+08	363451
Izz	0	0	4.16356e+08	301691
Ixy	0	0	1.37226e+08	99433.8
Iyz	0	0	-7.92274e+07	-57408.2
Izx	0	0	-1.7961e+08	-130146
Rx	0	0	47.6218	47.6218
Ry	0	0	73.7817	73.7817
Rz	0	0	67.2213	67.2213

The computed moment and product of inertia, and the radii of gyration will also be listed with respect to the centroid using the *specified* coordinate system as shown below:

Moments of Inertia and Radii of Gyration (w.r.t. CG's in Coord. Sys. [0])

	Length	Area	Volume	Mass
Ixx	0	0	4.23736e+07	30704
Iyy	0	0	6.55942e+07	47530.2
Izz	0	0	3.17729e+07	23022.7
Ixy	0	0	234.25	0.358757
Iyz	0	0	-205.045	-0.266214
Izx	0	0	9.18023e+06	6652.03
Rx	0	0	21.4448	21.4449
Ry	0	0	26.6813	26.6815
Rz	0	0	18.5696	18.5697

Similar to planar objects, there exists a set of mutually orthogonal axes through any given origin about which the products of inertia of a solid are zero. These axes represent the principal axes of the solid. The orientation of the principal axes can be found by setting the products of inertia about any axis to zero and solving for the direction cosine vectors. If one of the principal axes is known, then the orientation of the remaining axes can be found the method discussed for plane sections. The principal moments of inertia for the solid finite element model are shown below:

Principal Moments of Inertia (w.r.t. CG's):

	Length	Area	Volume	Mass
I1	0	0	6.55942e+07	47530.2
I2	0	0	4.76737e+07	34544.5
I3	0	0	2.64727e+07	19182.2

The corresponding direction cosine vectors of the principal axes with respect to the *specified* coordinate system are listed below:




Direction Cosine Vectors (w.r.t. Coord. System [0])

	Length	Area	Volume	Mass
a11	0	0	6.96051e-06	1.00935e-05
a21	0	0	1	1
a31	0	0	7.95191e-06	1.36022e-05
a12	0	0	0.866029	0.866028
a22	0	0	5.59952e-06	1.36757e-05
a32	0	0	0.499994	0.499995
a13	0	0	0.499994	0.499995
a23	0	0	3.40637e-06	6.73322e-06
a33	0	0	-0.866029	-0.866028




7 Loads and Boundary Conditions

Introduction

This chapter provides a summary of the commands for loads and boundary conditions enforcement in the Basic System. For more information on applying different types of loads, boundary conditions, constraint equations, coupling of nodes, coordinate systems, and other related information, please refer to COSMOS/M User Guide (V. 1).

Loads and Boundary Conditions in the Basic System

The types of loads you can apply on models in the Basic System can be categorized as:

- Mechanical loads
- Thermal loads
- Gravitational loads




Mechanical loads include imposed forces, pressures, moments, enforced displacements, etc., Gravitational loads include loads generated due to gravitational effects, inertial effects, and so on. Thermal loads include temperatures imposed on the nodes of a finite element model. Gravitational and thermal loads are known as special loads in COSMOS/M, and as such, they require the activation of a special flag under the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command. You may have more than one load case associated with all those load types.

The types of constraints you can apply as your model in the Basic FEA System (STAR and DSTAR) can be categorized as:

- Displacement boundary conditions.
- Coupling of degree of freedom between nodes.
- Coupling based on geometric entities such as point-to-point, point-to-curve or point-to-surface.
- Constraint equations by enforcing a mathematical linear relationship between different degree of freedoms.
- Automatic Rigid or Hinge connections at the interface between incompatible elements with mesh continuity at the interface.
- Rigid Bounding between any two intersecting geometries without having to preserve the element type compatibility or mesh continuity at the interface. (See chapter 1 and 3 for more detail on the Bond features).

The table below outlines the submenus required for applying loads and boundary conditions in the Basic System. The submenu in this table are found under the menu LoadsBC. For more information on how to use the commands in these submenus, please refer to Chapter 3, *Exploring GEOSTAR*, of the COSMOS/M User Guide (V. 1).




Table 7-1. Menu for Loads and Boundary Conditions Enforcement

Load/BC Type	Menu
Mechanical Loads	LoadsBC > STRUCTURAL > DISPLACEMENT* LoadsBC > STRUCTURAL > FORCE* LoadsBC > STRUCTURAL > PRESSURE*
Gravitational Loads	LoadsBC > STRUCTURAL > GRAVITY
Thermal Loads	LoadsBC > THERMAL > TEMPERATURE
Thermal Loads	LoadsBC > LOAD OPTIONS
Boundary Conditions	LoadsBC > STRUCTURAL > DISPLACEMENT LoadsBC > STRUCTURAL > COUPLING LoadsBC > STRUCTURAL > BONDING Analysis > STATIC > Static Analysis Options (Hinged/ Rigid Connection)
Support Reactions	Analysis > REACTION
Temperature Dependent Loads	LoadsBC > FUNCTION CURVE
DOF Reduction in Frequency Analysis (Guyan)	LoadsBC > STRUCTURAL > MASTER DOF

* Displacement, nodal forces as well as pressures may be defined in any coordinate system including cylindrical and spherical systems.

Multiple Load Cases in the Basic System

It is required in many cases to calculate the response of structures to different types of loading conditions separately. The stiffness matrix for linear problems is constant while the load vector varies from one loading condition to another. If loading conditions were to be treated as different problems, then most of the solution steps would be identical including the stiffness matrix calculation and its decomposition. It is therefore plausible to compute and decompose the stiffness matrix only once and use it to calculate the results under different loading conditions.

A loading condition in GEOSTAR is referred to as a *load case*. By default, load case number 1 is active and all defined loading will be associated with load case 1. Other load cases may be activated using the **ACTSET** (Control > ACTIVATE > **Set Entity**) command and specifying the LC argument for a load case. All loading conditions are associated with the active load case and will not affect other load cases. A load case defined through the **LCSET** (Analysis > STATIC > **Activate**

Load Case) command is referred to as a *primary* load case. Up to 50 primary load cases may be defined.

The principle of superposition states that the response of a linear structure to a loading condition defined by scaling (multiplying factors for loads) and combining two other loading conditions may be obtained by scaling and combining the results of the two loading conditions in the same way. The **LCCOMB** (Results > **Combine Load Case**) command in GEOSTAR is used to create a new load case and calculate its results by scaling and combining other load cases. Load cases defined through the **LCCOMB** (Results > **Combine Load Case**) command are referred to as *secondary* load cases. Up to 50 secondary load cases may be defined. The following commands are related to multiple load cases in GEOSTAR:

Table 7-2. Commands for Multiple Load Cases in GEOSTAR

Command (Cryptic)	Function
ACTSET	Activates a particular load case
LCCOMB	Defines a secondary load case
LCSET	Allows the user to select which defined load cases to solve and which to ignore, all primary load cases are considered by default
LCLIST	Lists defined load cases

While the loading varies from one primary load case to other, boundary conditions are usually kept the same. However, it is possible to combine load cases with different boundary conditions using the following procedure:

- Define one or more load cases with a set of boundary conditions.
- Use the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command to solve.
- Define a new set of boundary conditions.
- Define new load cases.
- Use the **LCSET** (Analysis > STATIC > **Activate Load Case**) command to deactivate the previously solved load cases.
- Use the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command to solve for the new load cases.
- Use the **LCCOMB** (Analysis > STATIC > **Combine Load Case**) command to combine results from the two sets as desired.

In using multiple load cases, users should be aware of the following:

- If GAP elements exist in the model, secondary load cases may be meaningless in most cases.
- If in-plane loading effects are considered, results are calculated for the active load case only.
- If buckling or nonlinear analyses is performed on a database with multiple load cases, then only the active case is considered.

The results of secondary load cases are not written to the output file, but a listing of the results may be obtained by using the **LISTLOG** (Control > MISCELLANEOUS > **List Log**) command and various on-line listing commands as desired.

Gravity, Centrifugal, and thermal loading are referred to as special loading in GEOSTAR and are only considered if the special loading flag is activated using the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command. Gravity and centrifugal loading as well as thermal loading can be included in any load case. Special loads are also associated with the active load case when they are defined and may be combined with other mechanical loads.

✍ Do ***not*** attempt to use multiple load cases when in-plane effects (differential stiffness) are activated for static, modal or buckling analysis. The program computes the results for the active load case only.

Multiple Thermal Load Cases

Thermal effects can be included in all of the 50 primary load cases in addition to other types of loading. In order for thermal effects to be considered in static analysis, the user has to activate the thermal loading flag using the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command. Thermal loading may be specified in two ways:

a. User-defined temperature profiles:

   Nodal temperatures are defined through commands in the LoadsBC > TEMPERATURE submenu. The defined temperatures will be associated with the active primary load case. The **TREF** (LoadsBC > LOAD OPTIONS > **Reference Temp**) command may be used to specify a reference temperature.

b. Temperature profiles read from heat transfer analysis:

   Temperature profiles from specified time steps are assigned to corresponding primary load cases through the **TEMPREAD** (LoadsBC > LOAD

OPTIONS > **Read Temp as Load**) command. A heat transfer analysis must be performed prior to running the static analysis on the same database using HSTAR or the FFE Thermal module. Time step number one should be used in the **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) command to assign a temperature profile from a steady state heat transfer analysis.

The following restrictions must be considered in using thermal loading:

1. If the user associates a thermal time step to a certain load case, then all user-defined nodal temperatures will be ignored for that load case.

2. You may use both types (a and b), mentioned above, in the same problem but in different load cases.

3. If you define temperature dependent material properties then the stiffness matrix is formed using the interpolated material properties based on the temperatures of the first considered load case in the analysis (i.e. load case with lowest label) regardless of the nature of that load case (i.e. whether it has a temperature profile or not).

4. If a temperature gradient is defined, then for the applicable elements, the temperature gradient will be applied to all load cases when the thermal loading flag is active.

5. Reference temperature applies only to the thermal load cases. A thermal load case has a temperature profile assigned.

6. Since buckling and other types of problems with geometric stiffness calculation are only considered for the active load case, therefore, only temperatures associated with the active load case will be considered in the analysis for those types of problems.

7. Use the **TEMPRDLIST** (LoadsBC > LOAD OPTIONS > **List Step/Load Case**) command to list assignments of temperature profiles from heat transfer time steps to primary load cases.

The TEMP1.GEO file in the example section illustrates the proper use of the commands pertinent to multiple thermal stress analysis.

8 Modeling and Analysis Guidelines

Introduction

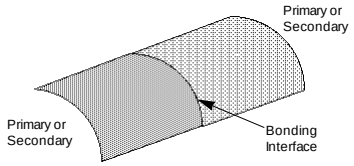
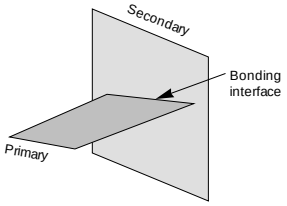
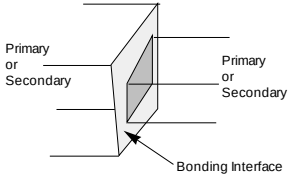
This chapter presents some modeling hints on some special topics in the Basic System. For more information on many other modeling topics, please refer to Chapter 3, *Exploring GEOSTAR*, of the COSMOS/M User Guide (V. 1).

Bonding of Meshes with Noncompatible Elements

The **bond** feature allows the user to connect finite element meshes between any two intersecting geometries *without* having to preserve the element type compatibility or mesh continuity at the interface. The geometric entities and corresponding element groups that can be bonded together are shown in the table below:




Table 8-1. Geometric Connections for Using Bond

Primary Entity	Secondary Entity	To connect...	Example
CR	CR	PLANE2D to PLANE2D SHELL to SHELL	
CR	SF or RG	SHELL to SHELL SHELL to SOLID	
SF or RG	SF or RG	SOLID to SOLID	

In the above table, SHELL refers to all 3-node triangular and 4- or 9-node quadrilateral shell elements that are supported in COSMOS/M. Similarly, SOLID refers to 8- or 20-node hexahedral solid elements as well as 4- or 10-node TETRA and 4-node TETRA4R solid elements. Some of the typical applications of the bond command are also shown in the above table.

✍ The bond feature is currently applicable to linear static, frequency, buckling, nonlinear structural, and heat transfer analyses.

The bond capability is specified using the BONDING submenu from LoadsBC > STRUCTURAL. The **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameters**) command bonds faces of elements associated with the selected geometric entities. The user specifies a primary bond entity (curve, surface, or region) and a pattern of target entities (curves, surfaces, or regions). All geometric entities must have been meshed before issuing this command in order to generate the bond information. Element edges/faces associated with the primary geometric entity are bonded with edges/faces of the secondary entities. The command is useful in connecting parts with incompatible mesh at the interface. Refer to the Command Reference Manual (Volume 2) or on-line help for details.

The **BONDLIST** (LoadsBC > STRUCTURAL > BONDING > **List**) command can be used to list a pattern of bond sets previously defined by the **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**) command. A typical listing is as follows:

```
 Set  Stype    Source   Ttype    #Targets  Targets
 1      CR        53      SF         1      7
 2      CR        50      SF         1      7
 3      CR        47      SF         1      7
 4      CR        44      SF         1      7
 5      CR        41      SF         1      7
```

Examples of Bond Connections

The following figures show examples of non-compatible connections where bond definition is required.

Figure 8-1. Solid-to-Shell Connection

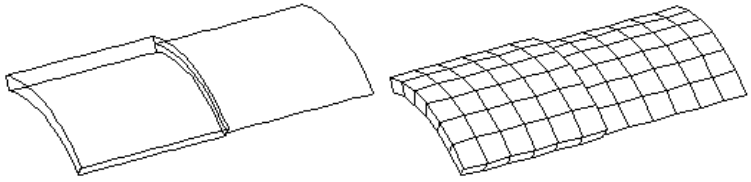


Figure 8-2. Shell-to-Shell Connection

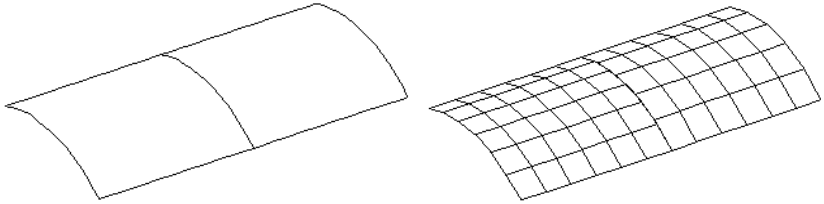


Figure 8-3. Solid-to-Solid Connection

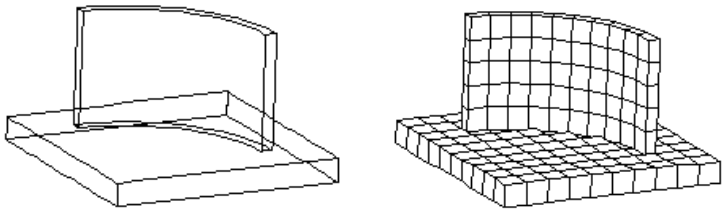


Figure 8-4. Shell-to-Shell Connection

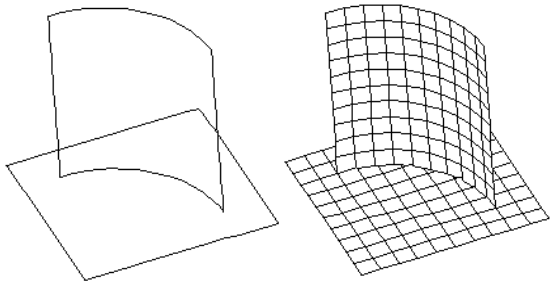
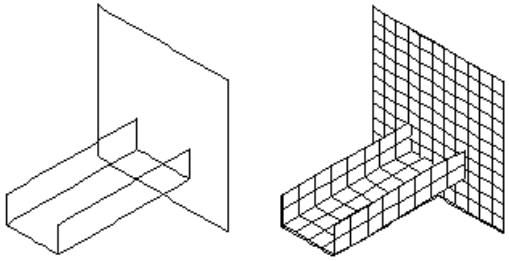


Figure 8-5. Shell-to-Shell Connection

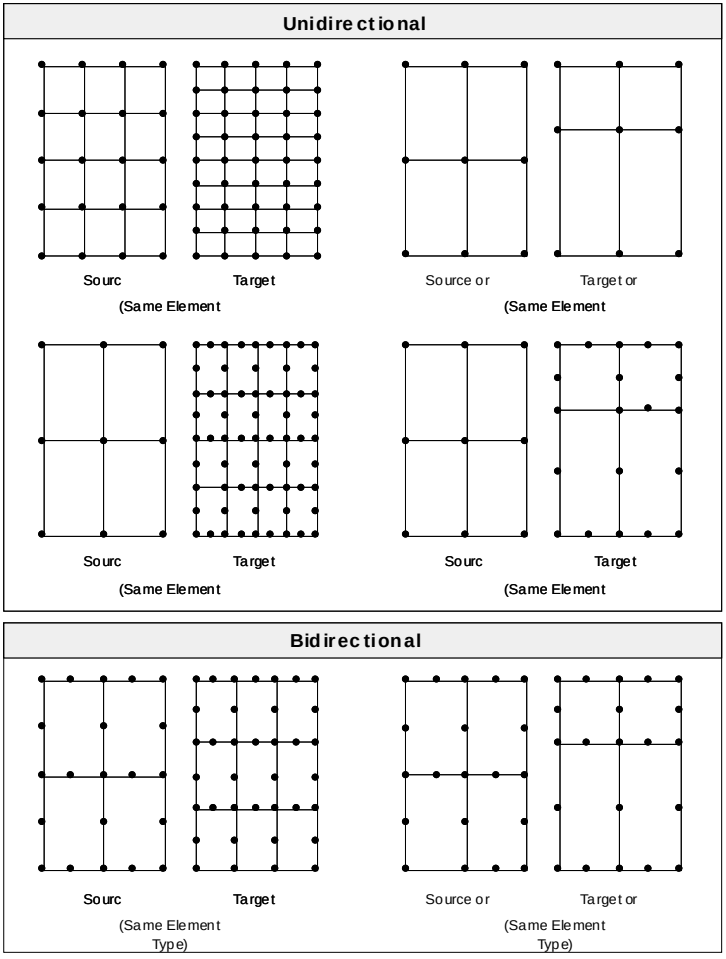


Guidelines for Using the Bond Capability

The following points should be considered in the application of this command:

- The **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameters**) command internally uses constraint equations to match the displacements and rotations of the two parts. The quality assurance tests have shown that for parts with reasonable stiffness properties and mesh densities, the maximum displacement and stress values obtained from the bond command are within ten percent of those values obtained from a merged model with compatible elements.
- This command is currently applicable to linear static and nonlinear structural analysis, frequency, buckling and advanced dynamics analysis as well as heat transfer analysis using HSTAR.
- The **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**) command offers the option of choosing between uni-directional bond (i.e. connecting all the nodes on primary entity to the elements on the secondary entity) or bi-directional bond (i.e. connecting the nodes on each entity to the elements on the other entity). The one directional bond should be used when connecting lower order elements of the primary (source) entity to lower or higher order elements of the secondary (target) entity. The bi-directional bond should be used in connecting higher order elements of the primary entity to higher order elements of the secondary entity. The following figure illustrates uni-directional and bi-directional bonding.

Figure 8-6. Uni-directional and Bi-directional Bonding Examples



For both types of bonding:

- Primary or the source entity is always the one that has fewer degrees of freedom
- Secondary or the target entity is always the one that has larger number

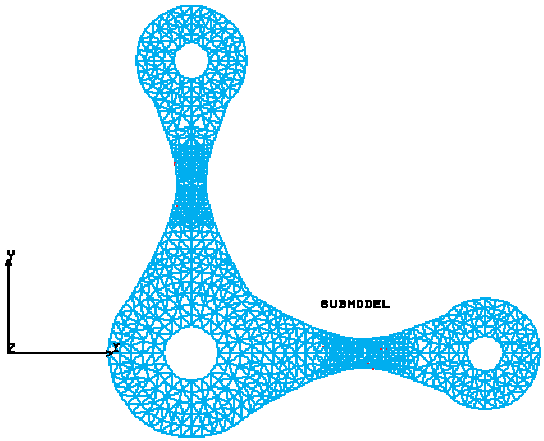
- When bonding solids and shells, it is advisable to use shells as the source and solids as the target irrespective of the element order.
- When shell elements are connected to solid elements, the common nodes at the boundary should ***not*** be merged as this will free the rotational degrees of shell at that node. Actually, it is advantageous ***not*** to have coincident nodes at all in such problems. In shell-to-shell, or, solid-to-solid connections, merging of the coincident nodes at the boundary ***is*** allowed.

- In problems where the stress concentration at the bonded intersection is critical, both parts should have a fine mesh in this region.
- The results obtained from the **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**) command may deteriorate in problems where a rigid part is connected to a relatively flexible part. The *bonded* area in the flexible part undergoes warping or has high displacement gradients. The results will improve if the mesh density for the flexible part is increased in the bonded area.
- The actual constraint relations between the nodes of source and target geometric entities are formed and computed in the analysis stage.

Submodeling

After you run your problem with a relatively coarse mesh, you may define the areas of concern as submodels where stresses may not have been calculated accurately due to sharp corners, geometric or load non-uniformity. The mesh inside the submodel will be refined and you may rerun the problem to improve results for the submodel only without changing results in the rest of the model. The submodeling process accelerates the job of obtaining accurate results in areas of stress concentration for large problems. With submodeling, you avoid the overhead of having to solve the whole model including areas of less importance at which a coarse mesh would have been sufficient to obtain acceptable results.

Figure 8-8. Typical Submodel with Refined Mesh



Submodeling Principles

Submodeling-is based on the St. Venant's principle. You may cut a portion of the model and run analysis only for the selected portion provided that displacements are properly prescribed at the boundaries of the cut. If displacements at the boundaries of the cut are calculated accurately at the first run, then these displacements may be considered as boundary conditions for the submodel run. The boundaries of the submodel must be adequately far from stress concentration areas.

Advantages of Submodeling

- For large problems (i.e., larger than 10,000 dof.), submodeling could save a lot of computer time and resources.
- You get accurate solutions at stress concentration areas.
- It allows you to repeatedly change or modify portions of the model during the design process.

Available Elements

The local mesh to be refined should consist of the following triangular shape elements.

- Linear TRIANG elements
- SHELL3, SHELL3T, and SHELL3L
- TETRA4 and TETRA4R

Available Features and Functions

1. Although the local mesh to be refined should consist of the elements mentioned above, the rest of the model may include any other types of elements.
2. You may deactivate submodeling at any time and run for the whole model.
3. Several disconnected portions of the model may be selected to create the submodel.
4. After you run for the submodel, you may define arbitrary new submodels.




5. The user need not be concerned about the nodal forces or element pressures within the submodel since the refinement is accomplished without moving the original nodes. In case of nodal temperatures however, the user may have to define temperature for the newly introduced nodes of the submodel.
6. You may use submodeling with multiple load case problems.
7. Displacement and stress fields outside of the submodel remain unchanged during the submodel run and may be integrated with the submodel during postprocessing by deactivating the element selection list.

Submodeling Steps

Submodeling is an extremely simple process. You only need to follow these steps.

1. Run the whole model with a relatively coarse mesh.
2. Create an element selection list using commands in the Control > SELECTION submenu.
3. Activate submodeling using the **SUBMODEL** (Analysis > STATIC > **Define SubModel**) command.
4. Run analysis using the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command.
5. Use postprocessing commands to display the results as usual. You may postprocess the submodel only or turn off the element selection list to postprocess the whole model.
6. Repeat the above steps as desired for new submodels if necessary. Refer to Chapter 11, *Examples*, for an example on submodeling.

Stresses in a Desired Coordinate System in Linear Static Analysis

In many cases, stresses are desired in a certain coordinate system. In COSMOS/M, stresses may be obtained in any defined Cartesian, cylindrical, or spherical coordinate system. It should be mentioned that the choice should be made before issuing the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**),

or the **R_STRESS** (Analysis > STATIC > **Run Stress Analysis**) commands. If displacements were already calculated, it is sufficient to choose the desired coordinate system and issue the **R_STRESS** (Analysis > STATIC > **Run Stress Analysis**) command. Two flags control the coordinate system that will be used for stress calculations for each element.

The first flag is an option in the **EGROUP** (Propsets > **Element Group**) command to specify whether stresses are desired in the local or the global coordinate system. The option is available for all 2D (TRIANG, PLANE2D, and all SHELL'S), and 3D elements (SOLID and all TETRA'S).

The second flag is an element attribute referred to as the element coordinate system (ECS) in the User Guide. Just like the element group, material property, and real constant sets, the ECS is listed for each element by the **ELIST** (Edit > LIST > **Elements**) command. To associate elements with a particular ECS, the **ACTSET** (Control > ACTIVATE > **Set Entity**) command may be used to activate the desired ECS before generating these elements. The ECS value for a pattern of elements may be modified using the **EPROPCHANGE** (Propsets > **Change El-Prop**) command. The default value for ECS is -1, which refers to the local element coordinate system defined by nodal connectivity as shown for each element group in Chapter 4, *Element Library*, of the COSMOS/M User Guide (V. 1)

The flag in the element group definition controls whether to use the assigned ECS values. If the global option (default) is used, then stresses are calculated in the Cartesian global coordinate system (system 0), regardless of the ECS values. If the local coordinate system is used, then control is transferred to the ECS values for all elements in that element group and the stresses are calculated in the specified ECS for each element. For shell elements, it is some times necessary to modify the ECS by projecting to the shell surface.

It should be remembered that the process of averaging the stresses is based on the values obtained in the specified coordinate system for each element. Element stresses at the center are calculated by averaging the stresses at the nodes of the element. Stresses at a node are calculated by averaging the corresponding values coming from all 2D and 3D elements to which the node is common. Mixed and meaningless results are generally obtained if averaging is performed in different coordinate systems. The fact that averaging of stresses is based on selection lists is helpful in cases where various ECS values are intentionally used for various elements. Selection lists may be defined using commands in the Control > SELECT menu.

Forces and stresses in 1D elements are always calculated in the ECS = -1 coordinate system and are not included when nodal stresses are calculated by averaging.




9 Performing Analysis

Introduction

Performing analysis for many finite element models is a very simple process in COSMOS/M. As you witnessed in the third chapter, whether your model is small or large, the preparation of input data for the finite element model and submitting it for analysis is set up in an efficient menu structure. This structure also allows you to perform different types of analysis on the same problem geometry without leaving GEOSTAR. In addition, the results of one analysis module can be transferred to another as loads for a multidisciplinary analysis (see Chapter 12). To accommodate the various needs of novice and experienced users alike, COSMOS/M provides a flexible set of tools to make the process of performing an analysis an efficient one.

This chapter explains the commands required for performing an analysis in the Basic System, with emphasis on specifying various options for analysis and output. For more information on commands mentioned in this chapter, please refer to COSMOS/M Command Reference Manual (Volume 2). This chapter also provides some common pitfalls you may come across while performing an analysis in the Basic System and suggests the required remedies.

Performing Analysis in COSMOS/M

The process of performing a finite element analysis using the Basic FEA System can be categorized into the following phases:

- Pre-analysis operations
- Specifying output options
- Transferring loads from multidisciplinary analyses
- Specifying analysis options
- Verification of analysis options
- Performing analysis

The commands required for executing the above phases is shown in the table below.

Table 9-1. Analysis Operations and Required Commands

Analysis Task	Required Commands (Cryptic) *
Pre-analysis operations	SHOW_MERGE, NMERGE, NCOMPRESS ECOMPRESS ECHECK DATA_CHECK R_CHECK
Specifying Output Options	PRINT_OPS PRINT_NDSET PRINT_ELSET
Transferring loads from other modules for multidisciplinary analyses (ESTAR, FLOWSTAR, and HSTAR)	EMFREAD NPRREAD TEMPREAD
Specifying analysis options	LCSET, LCLIST A_STATIC A_STRESS ADAPTIVE A_BUCKLING A_FREQUENCY
Verification of analysis options	A_LIST
Performing analysis	R_STATIC R_STRESS R_BUCKLING R_FREQUENCY

* Please refer to Appendix B, *Command Summary*, for locating the path to menus from cryptic commands.

Unless otherwise specified, all commands referred to in this chapter are found in the Analysis menu, OUTPUT OPTIONS, STATIC, and FREQUENCY/ BUCKLING submenus.

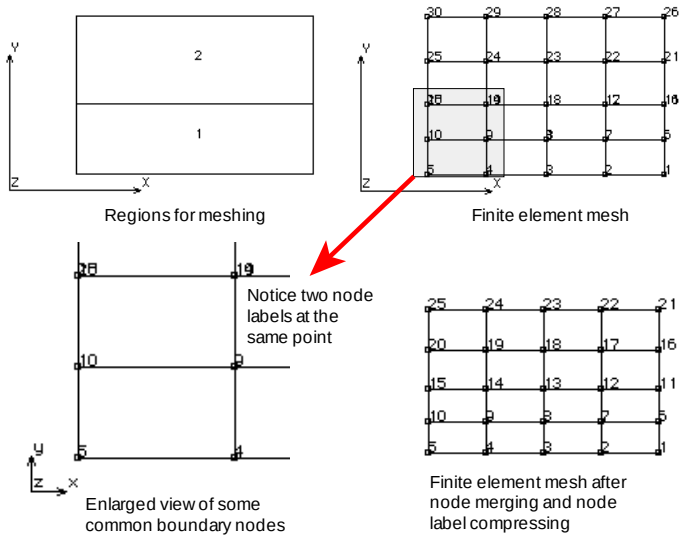
Pre-Analysis Operations

Pre-analysis operations refers to those operations you need to perform before submitting your model for analysis. These operations may include merging of nodes, checking the model data for errors, and specifying the options for renumbering of nodes and stress computations. The following paragraphs discuss these operations in more detail.

Compatibility Requirements

Since the mesh generation is performed independently for each region of your model, there will be duplicate definitions of nodes on common boundary curves and surfaces. Compatibility requirements dictate that the nodes on the common boundaries be merged in order to avoid problems in analysis (except for the crack or bound element). For example, consider the geometry shown in the figure below:

Figure 9-1. Merging of Nodes on Common Boundaries



The command **NIDENT** (Edit > IDENTIFY > **Nodes**) can be used to identify the generated nodes. At the common boundary, the node labels pointed by the arrow in the above figure are:

Control Panel: Edit > IDENTIFY > **Nodes**
Pick node id:19,37.5 25 0,
Pick node id:14,37.5 25 0,

You can notice that nodes 14 and 19 have the same x,y,z coordinates at (37.5, 25, 0), and one of these nodes needs to be merged using the **NMERGE** (Meshing > NODES > **Merge**) commands. The **SHOW_MERGE** (Meshing > NODES > **Show Merged Node**) command can be used to determine which nodes need to be merged. The use of the **NMERGE** (Meshing > NODES > **Merge**) command results in coinciding nodes (whose coordinates are within a specified tolerance) to be merged. You can merge selectively, or with respect to all nodes. The nodes are merged only if the differences between their corresponding X, Y, and Z coordinates are all less than the tolerance specified. The all/among flag specifies whether all the nodes in the model are to be checked for merging with the specified pattern of nodes, or to check merging only among the nodes specified in the pattern. The echo flag controls whether to give a message when two nodes are merged. The low/high flag determines which node to keep and which node to delete when two nodes get merged.

Element connectivity, load and boundary conditions are automatically updated after merging. If two nodes are merged and both have prescribed loads or boundary conditions, then the highest magnitude for each component is retained. If the loads or boundary conditions on the two merged nodes are specified in different coordinate systems, then the prescribed conditions for the *retained* node are enforced and the prescribed conditions for the deleted node are ignored.

If you plan to generate finite element meshes from existing meshed geometric entities by operations like extrusion, sweeping, flipping, moving, etc., then you are recommended (for proper association of the mesh with geometric entities) to postpone the use of **NMERGE** (Meshing > NODES > **Merge**) command until all such operations have been performed.

Since the merging operation deletes some nodes, the node numbers will not be consecutive. The **NCOMPRESS** (Edit > COMPRESS > **Nodes**) command which consecutively renumbers the nodes in the specified pattern can be used to remove the numbering gaps.

✍ In general, it is a good practice to use the **NCOMPRESS** (Edit > COMPRESS > **Nodes**) command so that the resulting consecutive node numbers are within the upper limit on the number of nodes.

✍ On a similar note, if you have deleted parts of your mesh, it is a good practice to use the **ECOMPRESS** (Edit > COMPRESS > **Elements**) command so that the resulting consecutive element numbers are within the upper limit on the number of elements.

Model Verification

To provide more flexibility's to the novice and experienced users alike, COSMOS/M features a hierarchic approach to checking the finite element model data before it can be submitted for analysis. You can verify the validity of either selected elements or the entire finite element mesh. Further, you can also verify if the *complete* finite element model is sound.

The **ECHECK** (Meshing > ELEMENTS > **Check Elements**) command checks the aspect ratio for the elements specified in the pattern. A message is issued if the aspect ratio of an element exceeds the specified value. The command automatically deletes degenerate elements from the database. It is recommended to issue this command before running the desired analysis.

The **DATA_CHECK** (Analysis > **Data Check**) command verifies that an element group, a material property set and a real constant set (if needed) have been defined for each element in the database. GEOSTAR automatically constrains any extra degrees of freedom (dof). This command does not constrain any dof. It is in general recommended that the **DATA_CHECK** (Analysis > **Data Check**) command be issued prior to any solution step.

✍ The **DATA_CHECK** (Analysis > **Data Check**) command is a subset of the **R_CHECK** (Analysis > **Run Check**) command which performs many other checking operations. Therefore, for a thorough check on the finite element model developed, you should consider using the **R_CHECK** (Analysis > **Run Check**) command.

The **R_CHECK** (Analysis > **Run Check**) command performs a thorough check on the database of the problem at hand and writes a report on the status of the input in a file, *probname*.CHK.

The following checks are performed:




1. Performs all functions of **DATA_CHECK** (Analysis > **Data Check**), namely checking that there is an element group, a material property set and a real constant set associated with each element.

2. Issues a warning message if a non existing node is used to define an element.

3. For PLANE2D and all shell elements, a warning message is issued in the following cases:
   a. If the aspect ratio for an element exceeds 5.0.
   b. If an angle in a 3-node element is less than 20 or more than 135 degrees.
   c. If an angle in a 4-node element is less than 45 or more than 135 degrees.

4. For solid elements, only the aspect ratio is checked.

5. For BEAM3D and BOUNDARY elements and all triangular shell elements, the element connectivity is checked and a warning is issued if the area defined by the three nodes is less than 1.0E-15.

6. It is recommended that this command be always issued before running any analysis.

A typical output from **R_CHECK** (Analysis > **Run Check**) is as follows:

```
Output from CHECK program

Maximum node label     = 63
Number of nodes        = 63
Maximum element label  = 48
Number of elements     = 48
Check all elements (Aspect ratio, Material properties
Connectivities, Real constants, . . .)

Total number of elements checked  = 48
Total warnings found                0
Total errors found                  0
Checking completed
```

Enabling/Disabling Stress Calculations

The **STRESS** command sets up the calculation of stresses. By default, the computation of stresses is always on. However, for large problems, it is a good idea to run only the displacement solution and verify the solution before proceeding to stress calculations.

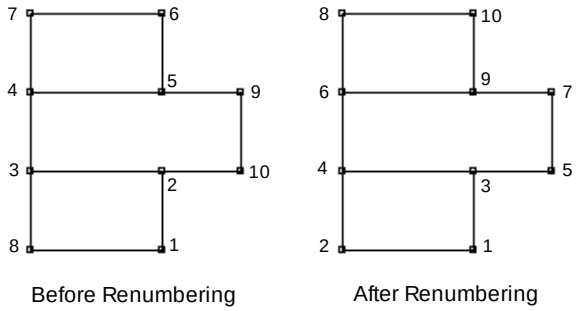
Node Renumbering

The **RENUMBER** (Analysis > **Renumber**) command renumbers the nodes to minimize the profile and bandwidth of the stiffness matrix. The **RENUMBER** (Analysis > **Renumber**) command only reads and stores the assigned value of the flag argument. Actual renumbering is done in the analysis program.

If you do not use this command, the program proceeds with node renumbering.

When you execute any analysis command, renumbering (module RENUM) is the first operation performed, followed by input processing (module PRE1) and the respective analysis modules. The summary of renumbering is written to a file, jobname.RNM.

Figure 9-2. Model for Renumbering



The renumbering summary is printed in the jobname.RNM file as follows:

```
Number of nodes     = 10
Number of elements  = 12

Before resequencing - - -
  Node difference          8
  Profile                  37
  Average node difference 3.70
  RMS node difference     3.96

After resequencing by GPS
  Node difference          3
  Profile                  28
  Average node difference 2.80
  RMS node difference     2.90

  1     1     1
  ... ...
The time required to reduce the Bandwidth is    1 Seconds
```

Specifying Output Options

The **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command controls the printing of displacements, velocities, accelerations, mode shapes, stiffness matrix, temperatures, temperature gradients, heat flow values, and detailed input information in the output file. This command has options to control the printing of displacements, velocities, accelerations, mode shape coordinates, etc. by means of respective flags. Additionally, if you request for input print flag, the nodal coordinates, element connectivities, material and real constant sets, applied loads, etc., will be written to the output file.

Filtered Output

For large problems, the output file can be very large as the results are written for all nodes and elements by default. However, before executing the analysis, you can specify the node and/or element numbers so that the output is generated for only the specified nodes and elements. The **PRINT_NDSET** (Analysis > OUTPUT OPTIONS > **Set Nodal Range**) command defines groups of nodes for which displacements, velocities and accelerations will be written in the output file (used with STAR and NSTAR modules only). Up to 10 groups can be specified.

The **PRINT_ELSET** (Analysis > OUTPUT OPTIONS > **Set Element Range**) command defines groups of elements for which stresses will be written in the output file (used with STAR and NSTAR modules only). Up to 10 groups can be specified.

Reaction Forces

The **REACTION** (Analysis > **Reaction**) command requests the calculation of reaction forces and/or moments at all constrained d.o.f.'s. Reaction forces are only calculated at constrained degrees of freedom. By default the reaction force is calculated for all constrained d.o.f's. Use this command only when you do not want to calculate the reactions.

Grid Force Balance

You may request Grid Force Balance at each node using a flag in the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command. STAR calculates and lists the forces from all elements sharing the node. The forces at each free node should be in equilibrium. However, at constrained nodes, these internal forces should be balanced by reaction forces. If reaction force is not requested with the grid force balance, then reactions are the opposite of the out-of-balance forces. Use the **DISLIST** (Results > LIST > **Displacement**) command to list reaction forces.

Transferring Loads for Multidisciplinary Analyses

The commands **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**), **NPRREAD** (LoadsBC > LOAD OPTIONS > **Read Fluid Pressure**), and **EMFREAD** (LoadsBC > LOAD OPTIONS > **Read Emag Force**) can be used to transfer loads from the HSTAR, FLOWSTAR and ESTAR modules of COSMOS/M. Chapter 12 presents detailed procedures for performing multidisciplinary analyses with examples.

The **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) command associates nodal temperatures from a steady-state or transient heat transfer analysis at a particular time step to a linear static analysis load case. If the type of heat transfer analysis performed is steady-state, then you need to associate time step number 1 to the desired load case.




✍ The heat transfer analysis must have been successfully completed before issuing the above command. The special thermal loading flag must be activated using the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command in order to consider thermal effects.

The **EMFREAD** (LoadsBC > LOAD OPTIONS > **Read Emag Force**) command is used for magneto-structural coupling, it provides the structural module with the magnetic nodal forces. You need to specify the time step number (in case of transient magnetic analysis) at which nodal magnetostatic forces are to be transferred for elastostatic analysis. If the type of magnetic analysis performed is steady-state, then you should specify 1 for this flag.

✍ Magnetic analysis with active force calculation option should be performed before using this command.

The fluid pressures computed after performing an analysis in FLOWSTAR can be transferred to the Basic System as pressure loading on elements. The **NPRREAD** (LoadsBC > LOAD OPTIONS > **Read Fluid Pressure**) applies the pressures calculated at any time step on the faces of elements associated with the specified curves. When you use this command, you will be required to provide information on the type of geometric entity at the transfer surfaces and their label numbers. Currently, only the curve entities are supported.

✍ The **NPRREAD** (LoadsBC > LOAD OPTIONS > **Read Fluid Pressure**) command is currently applicable for 2D problems only.

Specifying Analysis Options

The **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command specifies details of the linear static analysis to be later performed by the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command. The **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) command specifies details of the stress analysis to be later performed by the **R_STRESS** (Analysis > STATIC > **Run Stress Analysis**) command. The **A_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Buckling Options**) command specifies details of the buckling analysis to be later performed by the **R_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Run Buckling**) command. The **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command specifies details of the natural frequency analysis to be performed by the **R_FREQUENCY** (Analysis >

FREQUENCY/BUCKLING > **Run Frequency**) command. Please refer to the Command Reference Manual (V. 2) for more information.

Specifying Adaptive FEA Options

The **ADAPTIVE** (Analysis > STATIC > **Adaptive Method**) command specifies adaptive meshing for linear static problems. The parameters of this command are used by the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command to progressively improve the mesh until a desired accuracy level is reached. Mesh improvement is based on an estimate of the error in the strain energy norm using results from stress analysis. The highest errors are usually associated with elements at locations of stress concentration and elements with high aspect ratios.

Three methods are available for mesh improvement:

1. The H-method: Elements with high relative error are progressively subdivided until the error in the von Mises stress for each element approaches the average error within the specified tolerance.

2. The P-method: The polynomial order is increased from the present order to the specified one. This is done internally without introducing any new nodes in the database. The number of elements remains unchanged.

3. The HP-method: This method combines the H- and the P-methods. The H-method is progressively used until the error criterion is satisfied or the maximum number of loops is reached. After that the P-method is used to solve the refined mesh using the specified polynomial order.

Please refer to the Command Reference Manual (V. 2) for more information.

Specifying Load Case Options - Multiple Load Steps

The **LCSET** (Analysis > STATIC > **Activate Load Case**) command activates or deactivates specified primary load cases for static analysis. The **LCLIST** (Analysis > STATIC > **List Load Case**) command lists the type, number and run flag for the defined load cases.




Verification of Analysis Options

The **A_LIST** (Analysis > **List Analysis Option**) command lists the detailed options specified for various analyses. The analysis types supported by this command are shown below:

Type of Analysis	"type" Flag
Static analysis (*default*)	STATIC
FFE Static analysis	FFESTATIC
Stress analysis	STRESS
Frequency analysis	FREQ
Buckling analysis	BUCK
Fatigue analysis	FATIGUE
Nonlinear analysis	NONLIN
Thermal analysis	THERMAL
FFE Thermal analysis	FFETHERMAL
Fluid Flow analysis	FLOW
Electromagnetic analysis	EMAGNETIC
Optimization analysis	OPTIMIZE
Sensitivity analysis	SENSITIVITY
High frequency electromagnetic analysis	HFREQUENCY

Performing Analysis

The **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command performs linear static analysis. It calculates nodal displacements using the STAR program. Upon a successful run, the command, automatically runs the STRESS program to perform stress calculations unless otherwise specified by the **STRESS** (Analysis > STATIC > **Stress Analysis Options**) command. The **STRESS** (Analysis > STATIC > **Stress Analysis Options**) command sets the flag for stress calculations. The flag is used by the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command.

✍ By default, the program *always* computes stresses. Stresses are computed for all primary load cases that are set to run.

The command **R_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Run Buckling**) is used for performing eigenvalue buckling analysis. This command uses flags specified by the **A_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Buckling Options**) command.

The **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) command performs natural frequency analysis. It computes the frequencies and mode shapes using the DSTAR program. This command uses flags specified by the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command.

Analysis Problems

The following is a *partial* list of possible errors and recommended remedial procedures when you perform analysis using the Basic System. This list is by no means complete, and has been provided to supplement your engineering judgment. For more comprehensive list of errors, refer to the Trouble Shooting section of this manual.

Problem: Crashing Before Assembling the Stiffness Matrix

a. You may not have a finite element mesh. You can use the **NLIST** (Edit > LIST > **Nodes**) and **ELIST** (Edit > LIST > **Elements**) commands to make sure the nodes and elements have been properly created.

b. You may have zero degrees of freedom. This may result as a consequence of applying constraints on *all* nodes of the model. Check to see if the model has unconstrained degrees of freedom by using the **DLIST** (LoadsBC > STRUCTURAL > DISPLACEMENT > **List**) command. For large size problems, you may request for a detailed print out by activating the input print flag in the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command.

c. You may have a corrupted database. Create a new problem and read in the session file of the corrupted database problem using the **FILE** (File > **Load...**) command.

d. You may not have sufficient hard disk space available. The program warns you of this situation in the analysis screen.




Problem: Program Crashing During the Assembly of Stiffness Matrix

a. You may have degenerate elements. Run the **ECHECK** (Meshing > ELEMENTS > **Check Elements**) command to detect and delete these elements.

b. You may have inadmissible element group associated with some elements (for example, 4-node PLANE2D elements associated with a mesh of 3-node triangles generated by automatic meshing commands). Redefine the appropriate element group or change the mesh to the relevant type of elements by using **MASFCHANGE** (Meshing > AUTOMATIC > **Surface Mesh Type**) or other similar commands.

c. You may have very small element dimensions. Change the problem units.

d. You may have wrong element connectivity, especially for elements generated using the **EL** (Meshing > ELEMENTS > **Define Element**) command. Refer to Chapter 4, *Element Library*, in the COSMOS/M User Guide for the proper approach to generating element connectivity.

e. You may have two dimensional model in an inadmissible plane (for example, PLANE2D elements in X-Z plane).

Problem: Execution Abort Due to Singular Stiffness Matrix

a. You may have insufficient boundary conditions. Add constraints to prevent the possible rigid body motions.

b. You may have unmerged nodes. For models with multiple regions or surfaces, the nodes and elements are generated independently for each region/surface. However, compatibility conditions require that nodes on the common boundaries be merged. Use the **NMERGE** (Meshing > NODES > **Merge**) command with a proper merging tolerance.

c. You may have unconstrained nodes which are only attached to non-structural elements (e.g. CLINK, RLINK, MASS,…). Fix these nodes in all directions.

d. You may have incompatible nodes at the common boundaries. This may result from adjacent meshes with different element sizes and/or types. Regenerate the mesh with proper element sizes and connectivity or use the **BONDDEF** (LoadsBC > STRUCTURAL > **Define Bond Parameter**) command to bond dissimilar element meshes.

e. You may have exceeded the critical buckling load while the in-plane effect flag (see **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command) is on. Deactivate the in-plane effect flag or run a *nonlinear* analysis.




f. You may have a unstable structure due to constraining only the rigid body motion using GAP elements. Execute the problem with the soft springs option (see **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command) or add soft trusses on top of the gap elements.

g. You may have part of your axisymmetric model on the negative X-axis which is disallowed.

h. You may have defined physically inconsistent material properties for some elements (e.g. NUXY=0.6 or EX=0.0). Redefine these material properties.

i. You may have defined physically inconsistent material properties. In such cases, you need to define admissible values for all components of Poisson's ratio (e.g. NUXY, NUYX, NUXZ, NUZX, NUYZ, and NUZY). Redefine these material properties.

j. You may have wrong real constants values (for example, zero or negative thickness). *Note*: in case of singular stiffness matrix with zero diagonal term, run the static analysis with the soft spring option in **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command. Animate the deformation so that you can detect the rigid body motion direction. You can then proceed to constrain that direction.

Problem: Execution Abort During Stress Analysis

a. You may have run out of disk space (deactivate the stress print out flag).

b. You may have unrealistic input for the material properties or the real constants associated with the element causing the program termination.







10 Postprocessing

Introduction

Postprocessing in the Basic System is an efficient way to quickly analyze the analysis results for making design decisions. Postprocessing refers to the graphical manipulation of results in an organized and manageable form for easy interpre-tation. Finite element analysis results typically produce voluminous data, normally written to an output file. By allowing you to selectively choose data for either graphical processing on the screen or ascii listing on the screen, COSMOS/M provides one of the most powerful postprocessing features for finite element analysis results.

Features like sectioning, isoplots, and line section plots will assist you in better analyzing the results graphically on-screen. This chapter presents some examples of using these features.

Please refer to the COSMOS/M User Guide (V.1) for complete information on postprocessing in GEOSTAR, including hardcopy printing and plotting.

Postprocessing

The main postprocessing operations in the Basic System are: plotting and listing the desired component of analysis results. The following table lists the different postprocessing functions and the required commands. The commands in this

table can be found in the Results menu and ACTIVATE, PLOT, LIST, and EXTREMES submenus.

Table 10-1. Postprocessing Commands for Analysis Results in the Basic System

Postprocessing Operation	Command (Cryptic) *
Preliminary postprocessing operations	LCCOMB AVERAGE RESULTS?
Activating the type of analysis for postprocessing	ACTPOST
Activating the component for plotting or listing	ACTSTR ACTSTN ACTDIS ACTUSRPLOT
Plotting the activated component	IDRESULT ANIMATE DEFPLOT DISPLOT STRPLOT STNPLOT SMPLOT ISOPLOT SECPLOT LSECPLOT USRPLOT
Listing the activated component	DISLIST STRLIST STNLIST SMLIST BEAMRESLIST GAPRESLIST FREQLIST
Listing the extreme values of activated component	DISMAX STRMAX STNMAX SMMAX BEAMRESMAX
Setting on-screen plotting options	SETPLOT SETLSECPLOT SETERASE
Printing/plotting to hardcopy devices	see Vol. 1
Saving graphical results in various image file formats	see Vol. 1

* Refer to Appendix B, *Command Summary*, for locating the path to menus from cryptic commands.

✍ Results from an advanced dynamics analysis performed in the ASTAR module may be combined with the results of static analysis. This function is performed by the **READ_PDRESP** (Results > **Read Post-Dyn Response**) command.

The procedure for postprocessing is quite simple in COSMOS/M: perform the required preliminary operations such as combining load cases, etc., activate the required component of analysis result, and either plot or list for easy on-screen interpretation of analysis. Since most of postprocessing is a GEOSTAR function, it will not be attempted to list all details of postprocessing in this chapter. You can refer to Chapter 3, *Exploring GEOSTAR*, of the COSMOS/M User Guide (Volume 1) for more information. In the following section, some of the new postprocessing features will be discussed.

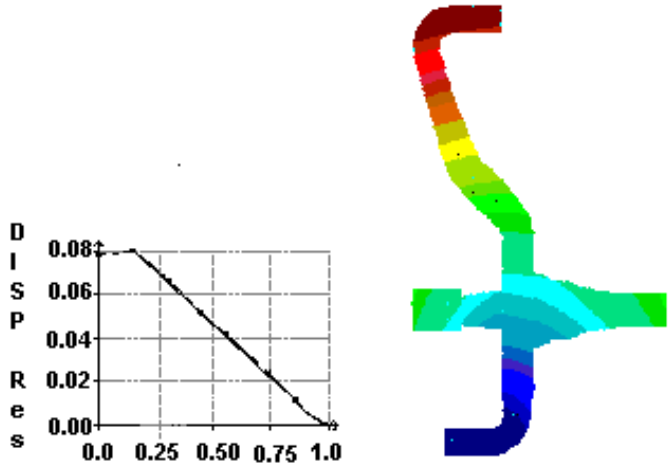
Plotting of Line Sections, Isoplanes and Cross Sections

It is now possible to graphically view the variation of a result component along a section line or a section plane using the commands **LSECPLOT** (Results > PLOT > **Path Graph** and **SECPLOT** (Results > PLOT > **Stress** or **Displacement**) with Section Plot option. You can also plot planes of same stresses or displacements for 3D models (isoplanes) using the command **ISOPLOT** (Results > PLOT > **Stress** or **Displacement**) with Isoplot option. These commands are applicable for any contour filled plot in 2D and 3D space.

The **LSECPLOT** (Results > PLOT > **Path Graph**) command generates an xy-plot from a filled contour plot in the active window. A path is specified by a number of nodes. The x-axis is used for the distance between nodes on the path and the reference node. The y-axis is the value of the plotted contour data. Two to twenty nodes can be used to define the path, and linear interpolation is used to calculate data along the section. For the following illustration, a line section plot was drawn by tracing a polyline through the cross section of the knuckle joint from top to bottom.

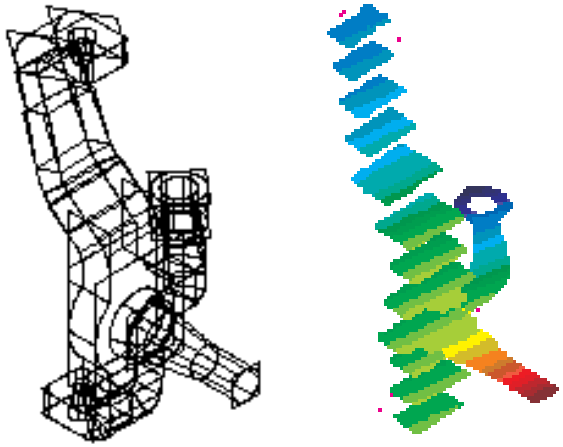



Figure 10-1. Line Section Plot of the Resultant Displacements in 2D Space



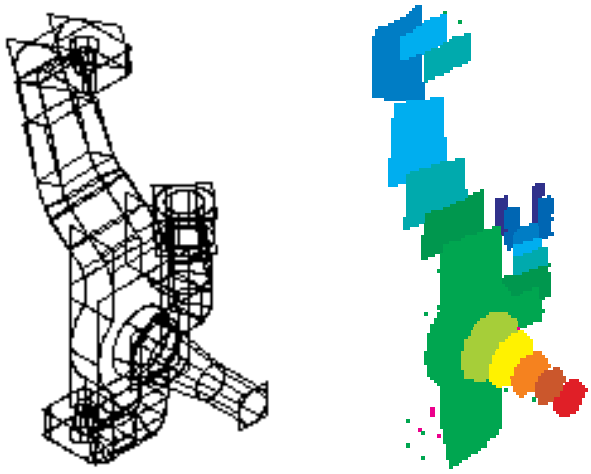
For the same knuckle joint, the **SECPLOT** (Results > PLOT > **Stress** or **Displacement**) command was used in 3D space to plot 12 sections of resultant displacement contour plot. The figure below shows the geometry as well as the section plot.

Figure 10-2. Cross Section Plot of the Resultant Displacements in 3D Space



The knuckle joint model was also analyzed with a plot of equal displacement contours. The **ISOPLOT** (Results > PLOT > **Stress** or **Displacement**) command plots isosurfaces for the active quantity in the plot buffer. The command works for 3D models only. A similar plot of isolines can be obtained for 2D models using the line option in contour plots. The figure below shows the geometry of the knuckle joint as well as the isoplots of resultant displacement.

Figure 10-3. Isoplots of the Resultant Displacements in 3D Space






11 Modeling and Analysis Examples

Introduction

This chapter presents some examples on the analyses you can perform in the Basic System. These examples cover detailed discussions of linear static analysis, buckling, and natural frequencies and mode shape computations. Almost all examples discussed in this chapter document commands required for model creation, mesh generation, load and boundary condition enforcement, analysis and postprocessing. Where possible, comparisons have been made with analytical solutions to indicate the accuracy of COSMOS/M.

To use these examples, enter GEOSTAR, and simply execute the commands as instructed. The computer prompts for commands are shown in 10 point Courier and the corresponding inputs are shown in bold, as illustrated below:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **EX: Elasticity modulus 1st dir**
Property value > **15.5E6**
Material Property Name >
(Click on cancel to terminate)

Please refer to the COSMOS/M User Guide (V. 1) for more information on using GEOSTAR.

Input files for the examples discussed here are available in the PROBLEMS.EXE file installed in the COSMOS/M directory (Macintosh and Unix Workstation versions may have a separate folder or directory with a title Example Problems or PROBS, respectively). On PCs, the file PROBLEMS.EXE can be unpacked by typing the command **PROBLEMS** at the DOS prompt. For other machines, follow the instructions provided to unpack or expand the archived file.

The second part of this manual presents many verification problems on all analysis features of the Basic System. You are recommended to use these examples to learn more about the Basic System.

Table 11-1. List of Modeling and Analysis Examples

List of Examples	
Contact/Gap	LGAP1 - Hertzian Contact Using Node-to-Node Linear Gap Elements.
Bond	BOND1 - Static Analysis of a U-Frame with Non-compatible Elements.
Linear Elastic Fracture Mechanics Analysis	LEFM1 - Stress Intensity Factor of a Plate with Two Edge Cracks.
	LFEM2 - Stress Intensity Factor of Three Point Bend Specimen.
Buckling Analysis	BUCKL1 - Buckling of a Rectangular Plate under Uniform Pressure.
Modal Analysis	MODAL1 - In-Plane Effects on the Natural Frequencies of a Column.
	MODAL2 - Modal Analysis of a Square Frame with Rigid Body Modes.
Submodeling	LSUBM1 - Using Submodeling for Shell Problems.
	LSUBM2 - Using Submodeling with Tetrahedral Elements.
	LSUBM3 - Using Submodeling for Shell Problems.
ASME Stress Requirement	ASME.GEO - ASME Code Stress Evaluation.
Use of Multiple Thermal Load Cases	TEMP1.GEO - Multiple Thermal Load Cases

Examples on Linear Static Stress Analysis

The types of linear static problems you can analyze in the Basic System include models with various combinations of mechanical, thermal, and gravitational loads, cracks, gaps and contact surfaces, substructuring, cyclic symmetry, and many others. The examples discussed in this section only address some features of the linear static analysis module STAR owing to space limitations. The verification problems presented in the second part of this manual cover almost all aspects of linear static modeling and analysis.

Example on Contact

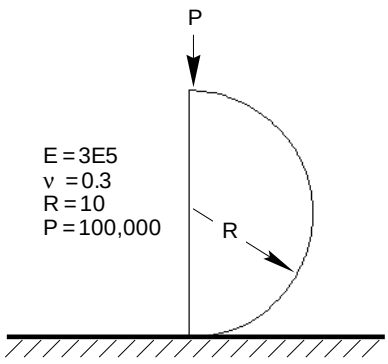
Contact problems are best solved as nonlinear problems. The example in this chapter shows the procedure to solve a simple contact problem using linear analysis. It should be noted that the principle of superposition, valid for regular analysis, may not hold when using linear analysis to solve contact problems.

Example LGAP1 - Hertzian Contact Using Node-to-Node Linear Gap Elements

The *complete* input for this example is available in the file LGAP1.GFM. The study of stresses caused by the pressure between elastic bodies are important in the design of train tracks, ball and roller bearings, expansion joints for steel girders, and many

other practical problems. Due to the stresses developed at contact areas which may be as high as the yield limit, the study of contact problems is often analyzed as a nonlinear finite element problem. However, a good approximation for the contact stresses can be obtained using the linear node-to-node contact/gap elements in the Basic System. The figure below shows the half model of a cylinder for contact stress analysis using node-to-node gap elements. The

Figure 11-1. Problem Geometry for Analysi



Half model of a cylinder on a flate plate (Hertz contact problem)

material properties, dimensions and applied load are also shown in the figure.

Since the complete input for this example is available to you, there is ***no*** need to issue any command for generation of the finite element mesh. The only two commands you need are the **FILE** (File > **Load...**) command to load the input file and the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command to execute analysis. In what follows, you will be presented with information on how the finite element model was developed for analysis. Where ever a command is required to be executed, you are clearly instructed to do so.

To start with, copy the file LGAP1.GFM to your working directory, enter GEOSTAR, and execute the **FILE** (File > **Load...**) command as illustrated below:

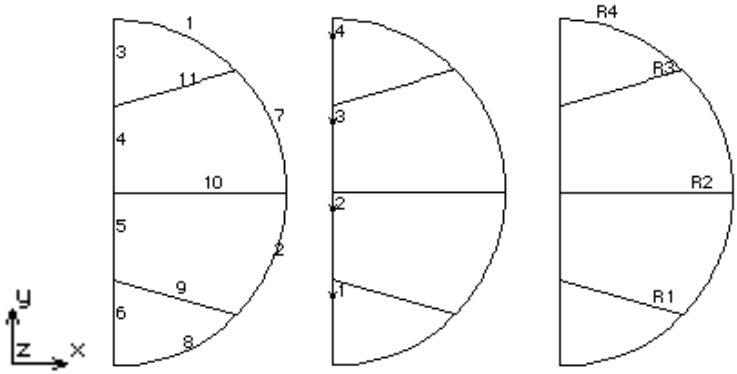
Geo Panel: File > **Load...**
Input file name with extension > **LGAP1.GFM**
Accept Defaults

The file LGAP1.GFM includes information on the problem geometry such as points, curves, contours, and regions as well as the finite element data such as nodes, elements, properties, loads, and analysis specifications. You will see the finite element model constructed and displayed as the input is read in. To properly see the model, use the Perfect Scaling icon, and the View icon to set the X-Y view.

The command **ACTNUM** (Control > ACTIVATE > **Entity Label**) can be used to activate numbering of geometric or finite element entities to see the respective labels generated. The geometry of the cylinder was generated by using the command **CRARC3PT** (Geometry > CURVES > CIRCLES > **by 3 Points**) for the curved part and **CRLINE** (Geometry > CURVES > **Line with 2 Pts**) for the straight line. These curves were further broken using the command **CRBRK** (Geometry > CURVES > MANIPULATION MENU > **Break (equally)**) to smaller segments to construct regions for mesh transition from coarse to fine. The figure below shows the curve, contour, and region labels of the cylinder. The list commands such as **PTLIST**, **CTLIST**, **RGLIST**, (Edit > LIST > **Points**, **Contours**, **Regions**, **...**) etc., can be used to list the created geometric entities.




Figure 11-2. Curve, Contour and Region Plots of the

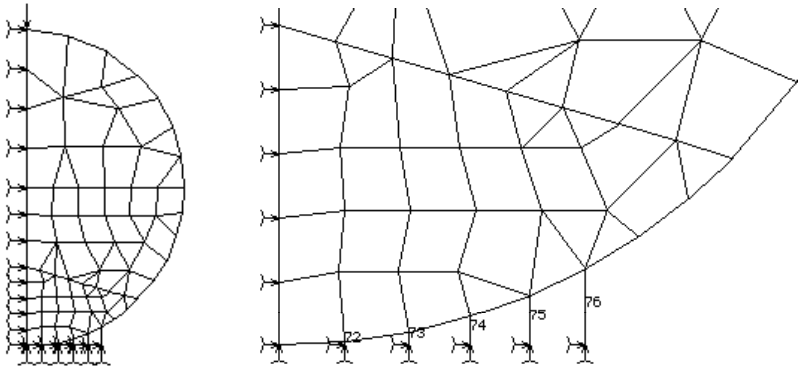


There are two types of elements used in this model. The PLANE2D element models the continuous domain of the cylinder in a state of *plane strain* whereas the GAP element was used to connect the cylinder to the contact surface. These elements were defined using the command **EGROUP** (Propsets > **Element Group**). The PLANE2D element was defined under group 1 and the GAP element was defined under group 2. Since the PLANE2D element models plane strain behavior, there are no section constants required for this element under real constant set 1.

The finite element mesh of the cylinder was generated using the command **MA_RG** (Meshing > AUTO MESH > **Regions**) for all four regions. The command **NMERGE** (Meshing > NODES > **Merge**) was used to merge the nodes at the common boundary curves. The GAP element is a *special* element which needs to be created by properly activating the required real constant sets as explained in the following paragraph.

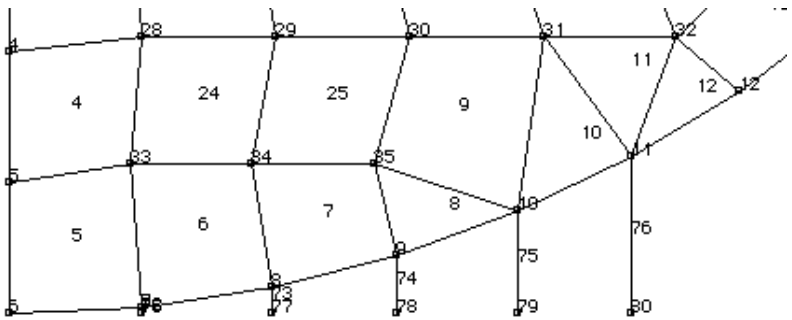
The figure below shows the complete finite element model of the cylinder with the applied loading and constraints. The GAP element was defined by using the **EL** (Meshing > ELEMENTS > **Define Element**) command which connects the specified nodes defined by the **ND** (Meshing > NODES > **Define**) command. In the Basic System, the gap closure tolerance (gdist) is defined as a real constant parameter. Therefore, each gap element requires a real constant set which has to be activated before the element is defined. For this problem, five gap elements numbered 72 through 76 were used.

Figure 11-3. Finite Element Model of the



The figure below shows a clear view of the connection between the cylinder and the bottom contact surface. To start with, the element group 2 was activated for gap elements, and nodes 76 through 80 were defined at a y-coordinate of zero and x-coordinates equal to the *exact* coordinates of the respective nodes of the cylinder to which they are connected. It is important to specify the *exact* x-coordinates for the nodes of the gap element as a slight eccentricity will introduce lateral components of forces on the gap. In the next step, real constant 2 was activated and the first gap element (label 72) was defined using the **EL** (Meshing > ELEMENTS > **Define Element**) command. This element connects nodes 7 and 76. Similarly, the remaining gap elements were defined after activating the respective real constant sets.

Figure 11-4. Enlarged View of the Contact Surface



For a gap element, you need to specify the allowable relative motion between the two nodes beyond which the gap will resist compression or tension. For compression gaps, the most common situation is to specify the relative motion between the two nodes as equal to the initial distance between them such that they come into contact before resisting compression. Due to the amount of labor involved in inputting these distances for a large number of gaps, an option is provided in the element group definition (**EGROUP** (Propsets > **Element Group**) command) to instruct the program to calculate and use the initial distance between the nodes of each gap element. Refer to Chapter 4, *Element Library*, in the COSMOS/M User Guide. In this case there is no need to define the real constants associated with the gaps and any input will be ignored.

The bottom nodes of all gap elements were fixed using the **DND** (LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**) command. Node 6 which represents the node directly in contact with the bottom surface was also fixed using the same command. Along the vertical axis of symmetry, the symmetric boundary conditions (UX = 0) were imposed using the **DCR** (LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**) command. The force at the top was applied using the **FND** (LoadsBC > STRUCTURAL > FORCE > **Define Nodes**) command. There is only one material set for this problem, defined for the cylinder. The properties E and ν were defined using the **MPROP** (Propsets > **Material Property**) command. To check the accuracy of the solution, the computation of reaction forces was requested using the **REACTION** (Analysis > **Reaction**) command.

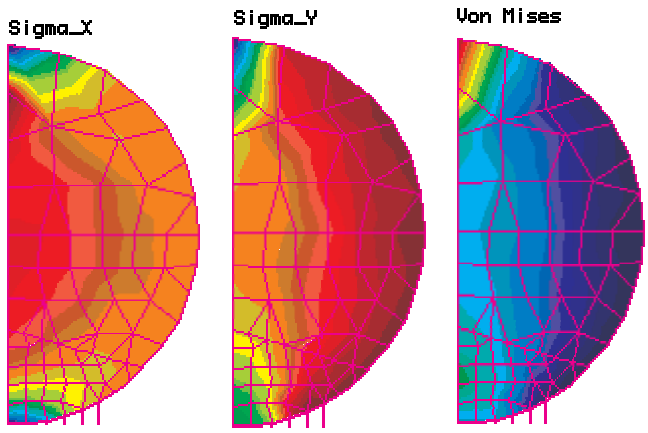
After you have inspected the model, *execute* the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) to perform linear static analysis. The output file was examined for reaction forces on the nodes in contact. As shown in the illustration below, the vertical component of reaction forces adds up to 100,000 which is the force applied, thus providing equilibrium check.




```
   LOAD CASE NUMBER. . . . . . . . . . . . =   1

   FORCES IN THE CLOSED BOUNDARY/GAP ELEMENT(S), NCE =     3
      ELEMENT     FX            FY            FZ       GAP DISTANCE   NORMAL FORCE
         72    0.0000E+00    0.4404E+05    0.0000E+00    0.5163E-01    0.4404E+05
         73    0.0000E+00    0.3153E+05    0.0000E+00    0.2032E+00    0.3153E+05
         74    0.0000E+00    0.6277E+03    0.0000E+00    0.4496E+00    0.6277E+03
      TOTAL    0.0000E+00    0.7620E+05    0.0000E+00

R E A C T I O N    F O R C E S

 NODE   CSYS      RFX          RFY         RFZ     RMX     RMY     RMZ
    6    0     0.1373E+05  0.2380E+05    ----    ----    ----    ----
```

To see the stress results, the required component of stress can be plotted using **STRPLOT** (Results > PLOT > **Stress**). The figure below shows the horizontal and vertical components of stresses as well as the von Mises stress contour plots.

Figure 11-5. Stress Contour



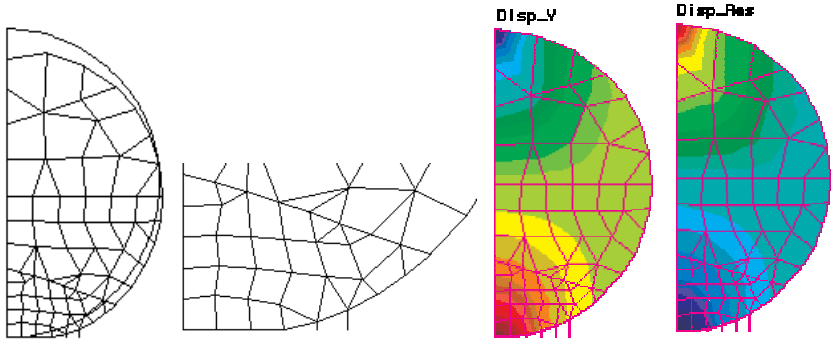
Hertz (1895) developed the mathematical theory supported by experimental results for surface stresses and deformations produced by pressure between curved bodies. The maximum compressive stresses according to his theory occur at the center of the contact surfaces while the maximum shear stresses occur at the interior of the compressed parts. The maximum contact stress is given by the following equation (Roark's Formulas for Stress and Strain, Sixth Edition, Young, 1989):

$$(\sigma_c)_{max} = 0.591\sqrt{\frac{pE}{D}} \qquad (11\text{-}1)$$

where p is the load per unit length and D is the diameter of the cylinder. Using the above equation, the vertical component of stress at the contact surface is 3.2376E6 whereas the maximum vertical component of stress *computed* at the center of element 5 is 4.43372E6. Because of the very small contact area in what one initially approximates as a line or a point, contact stresses for even lighter loads can be very high. Since in reality the stresses are highly localized and triaxial, the actual stress intensity can be very high, and this cannot be accounted for by linear elastic contact mechanics theories. Therefore the solutions provided by the finite element analysis can indeed be viewed as accurate.

The command **DEFPLOT** (Results > PLOT > **Deformed Shape**) can be used to see the deformed shape of the cylinder. The **DISPLOT** (Results > PLOT > **Displacement**) command can be used to view the displacement contours. The figure below shows the deformed shape (full and enlarged) as well as the vertical and resultant displacement contours of the cylinder model.

Figure 11-6. Deformed Shape and Displacement Contour



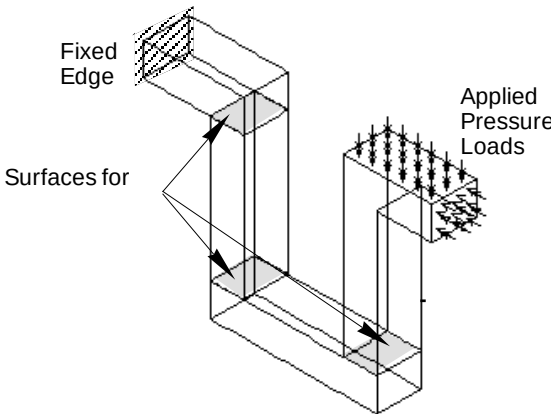
Example on Bond

There are many modeling problems in which the compatibility of elements and nodes at the common boundaries is often difficult to produce during mesh generation. As recommended in Chapter 8, *Modeling Guidelines*, the compatibility of elements must be always maintained, and only for exceptional cases, the bond feature can be used to tie the non-compatible parts of the mesh.

Example BOND1 - Static Analysis of a U-Frame with Non-compatible Elements

The input for the geometry of this problem including some geometric entities is provided to you in the file BOND1.GFM. In the descriptions to follow, the procedures for defining higher order geometric entities such as polyhedrons and parts, mesh generation, defining bonding surfaces, applying constraints and loads, and performing analysis are explained in detail. The model for analysis consists of a u-shaped cantilever solid subjected to pressure loading. The figure below shows the geometry of the model with applied boundary conditions and loads. The locations of the bonding surfaces are also indicated since the model will be meshed in such a way that the elements will be *mismatched* at these interfaces.

Figure 11-7. Model for Analysis Showing Locations for *Bond* Enforcement



There are about 4000 nodes, 2200 elements, and 11,700 equations in this example. Solution of linear static analysis requires about 30 Mb of free disk space on a IBM PC with 16 Mb of RAM. Make sure you have this disk space available on your computer before attempting to solve this example.

As you can notice, there are four solid volumes in this model. Half of the solid volumes will be meshed with hexahedral solid (SOLID) elements of different element size, and the remaining half will be meshed with tetrahedron solid (TETRA10) elements, also of different sizes. This process will result in a mesh with ***non-compatible*** SOLID-to-SOLID, SOLID-to-TETRA10, and TETRA10-to-TETRA10 element interfaces. *In a conventional finite element solver, this type of mesh will be unusable for analysis*. However, with the bond feature of COSMOS/M, all you need is the command **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**) to connect the non-compatible parts of the mesh together for providing compatibility, and proceed with the analysis. The bond definition process is performed at the geometric entity level such that the nodes and elements at the bonding interfaces are automatically identified and interconnected by means of constraint relations. Note that the term non-compatible here refers to the mismatching of nodes at the interfaces, and it is different from incompatible which refers to cases where the interface has matching nodes but the deformation fields of the elements are not compatible.

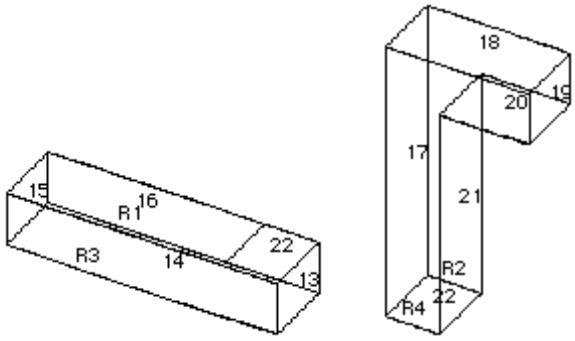
To start with, copy the file BOND1.GFM to your working directory, enter GEOSTAR, and execute the **FILE** (File > **Load...**) command as illustrated below:

Geo Panel: File > **Load...**
Input file name with extension > **BOND1.GFM**
Accept Defaults

You will see the geometry of the model constructed and displayed on the screen as the input commands are read. As you can notice from the geometry of the model, there are four solid volumes in this model. The geometry created includes curves, surfaces, regions, and two volumes. These two solid volumes created as volume entities will be meshed with brick elements using the parametric (mapped) mesh generation scheme (**M_VL** (Meshing > PARAMETRIC MESH > **Volumes**) command). The remaining two solid volumes are created by using polyhedron and part definitions and are subsequently meshed with tetrahedron elements using the **MA_PART** (Meshing > AUTO MESH > **Parts**) command.

We will proceed with the definition of polyhedrons and parts using existing surfaces and regions for 3D automatic meshing. The procedures for 3D automatic meshing is explained in COSMOS/M User Guide. In order to define a polyhedron, you need to first identify only those surfaces and regions that constitute the polyhedron. This is best achieved by making use of the selection feature available in GEOSTAR. The surfaces and regions constituting polyhedrons (also parts) 1 and 2 are shown in the figure below. Notice that surface 22 is common to both parts.

Figure 11-8. Regions and Surfaces for Part Entities 1 and 2



Before defining the first polyhedron, execute the **INITSEL** (Control > SELECT > **Initialize**) command as illustrated below separately for surfaces and regions:

Geo Panel: Control > SELECT > **Initialize**
Entity Name > **Surfaces**
Initialization Flag > **Yes**
Selection Set number > **1**

Entity Name > **Regions**
Initialization Flag > **Yes**
Selection Set number > **1**

As seen from the above figure, the first polyhedron is bounded by surfaces 13, 14, 15, 16, and 22, and regions R1 and R3. You need to place these entities in the selection list using the command **SELINP** (Control > SELECT > **by Label**) as illustrated below:

Geo Panel: Control > SELECT > **by Label**
Entity Name > **Surfaces**
Beginning Entity > **13**
Ending Entity > **16**
Increment > **1**
Selection Set number > **1**

Geo Panel: Control > SELECT > **by Label**
Entity Name > **Surfaces**
Beginning Entity > **22**

Ending Entity > **22**
Increment > **1**
Selection Set number > **1**

Geo Panel: Control > SELECT > **by Label**
Entity Name > **Regions**
Beginning Entity > **1**
Ending Entity > **3**
Increment > **2**
Selection Set number > **1**

If you clear the screen and issue **SFPLOT;** and **RGPLOT;** (Edit > PLOT > **Surfaces**, **Regions**) commands successively, you will see that only those surfaces placed in the selection list will be now plotted. You can now issue the **PH** (Geometry > POLYHEDRA > **Define**) command to define a polyhedron, and the program will only select the surfaces and regions in the selection list to form the entity. Specify an average element size of 5 for the first polyhedron as shown below:

Geo Panel: Geometry > POLYHEDRA > **Define**
Polyhedron Label > **1**
Reference entity name > **Surfaces**

Surface/Region Label > **1**
Average element size > **5**
Tolerance > **0.0001**
Redefine element size on bound > **Change size**

The polyhedron is a hollow entity, and it needs to be converted to a solid volume by using the **PART** (Geometry > **Define Part**) command as illustrated below:

Geo Panel: Geometry > **Define Part**
Part Label > **1**
Polyhedron 1 > **1**
Polyhedron 2 > **1**

Similarly, the surfaces and regions that constitute the second polyhedron are identified and placed in the selection list as illustrated below:

Geo Panel: Control > SELECT > **Initialize**
Entity Name > **Surfaces**
Initialization Flag > **Yes**
Selection Set number > **1**

Geo Panel: Control > SELECT > **Initialize**
Entity Name > **Regions**
Initialization Flag > **Yes**
Selection Set number > **1**

Geo Panel: Control > SELECT > **by Label**
Entity Name > **Surfaces**
Beginning Entity > **17**
Ending Entity > **22**

Geo Panel: Control > SELECT > **by Label**
Entity Name > **Regions**
Beginning Entity > **2**
Ending Entity > **4**
Increment > **2**
Selection Set number > **1**

Next, proceed to define the second polyhedron with an element size of 7.5 and the part entity as illustrated below:

Geo Panel: Geometry > POLYHEDRA > **Define**
Polyhedron Label > **2**
Reference entity name SF or RG > **Surfaces**

Surface/Region Label > **22**
Average element size > **7.5**
Tolerance > **0.0001**
Redefine element size on bound > **Change size**

Geo Panel: Geometry > **Define Part**
Part Label > **2**
Polyhedron 1 > **2**
Polyhedron 2 > **2**

To make all surfaces and regions available for other modeling purposes, you need to re-execute the **INITSEL** (Control > SELECT > **Initialize**) command for all entities as illustrated below:

Geo Panel: Control > SELECT > **Initialize**
Entity Name > **All**
Initialization Flag > **Yes**
Selection Set number > **0**




The procedures you executed so far focused on preparing the geometry for meshing. To reiterate, volumes 1 and 2 to be meshed with brick elements were predefined as volume entities and the command **M_VL** (Meshing > PARAMETRIC MESH > **Volumes**) can be directly applied on these volumes. The remaining two solid volumes were defined as part entities 1 and 2 for automatic mesh generation with tetrahedron elements using **MA_PART** (Meshing > AUTO MESH > **Parts**) command.

We will first generate the 8-node brick elements in volumes 1 and 2 using the mapped mesh generation scheme. Select the **M_VL** (Meshing > PARAMETRIC MESH > **Volumes**) command and specify input as illustrated below:

Geo Panel: Meshing > PARAMETRIC MESH > **Volumes**
Beginning Volume > **1**
Ending Volume > **1**
Increment > **1**
Number of nodes per element > **8**

Number of elements on 1st curve > **3**

Number of elements on 2nd curve > **10**

Number of elements on 3rd curve > **4**

Accept Defaults ...

The 8-node brick elements (SOLID) generated will be assigned with element group number 1 by default. Repeat the **M_VL** (Meshing > PARAMETRIC MESH > **Volumes**) command for the second volume with a different number of elements along each curve as illustrated below:

Geo Panel: Meshing > PARAMETRIC MESH > **Volumes**
Beginning Volume > **2**
Ending Volume > **2**
Increment > **1**
Number of nodes per element > **8**

Number of elements on 1st curve > **4**
Number of elements on 2nd curve > **15**
Number of elements on 3rd curve > **6**

Accept Defaults...




Before generating the tetrahedron elements in parts 1 and 2, you need to first activate a different element group number for proper association with the generated 10-node tetrahedron (TETRA10) elements:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set Label > **Element Group**

Element Set number > **2**

You can now proceed to generating the mesh for parts 1 and 2 using tetrahedron elements. Since surface 22 is common to both parts 1 and 2, and part 2 with element size 7.5 was last defined, the element density on surface by default will be 7.5. Therefore, before meshing part 1, you need to reset the element density on all surfaces of part 1 to 5 using the **PHDENS** (Meshing > MESH DENSITY > **Polyhedron Elem Size**) command as shown below:

Geo Panel: Meshing > MESH DENSITY > **Polyhedron Elem Size**
Beginning Polyhedron > **1**
Beginning Polyhedron > **1**
Increment > **1**
Average element size > **5**

Use the command **MA_PART** (Meshing > AUTO MESH > **Parts**) to generate 10-node tetrahedron elements as shown below:

Geo Panel: Meshing > AUTO MESH > **Parts**
Beginning Part > **1**
Ending Part > **1**
Increment > **1**
Hierarchy check flag > **Check Hierarchy**
Element order > **High**
Number of Smoothing Iterations > **4**

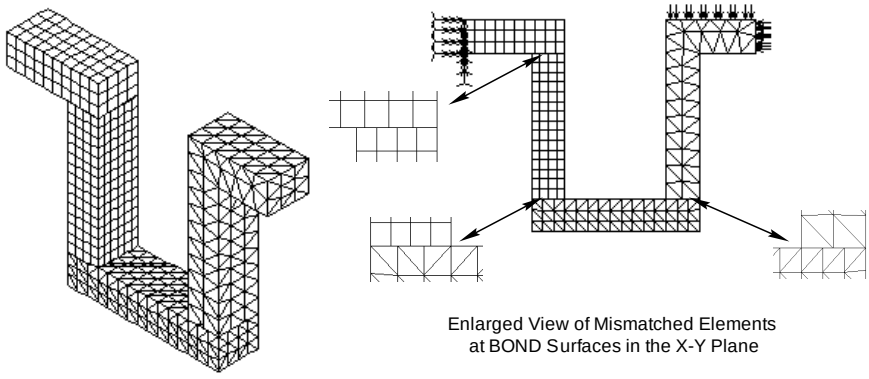
Repeat the **PHDENS** (Meshing > MESH DENSITY > **Polyhedron Elem Size**) command and mesh the second part with 10-node tetrahedron elements as shown below:

Geo Panel: Meshing > MESH DENSITY > **Polyhedron Elem Size**
Beginning Polyhedron > **2**
Beginning Polyhedron > **2**
Increment > **1**
Average element size > **7.5**

Geo Panel: Meshing > AUTO MESH > **Parts**
Beginning Part > **1**
Ending Part > **1**
Increment > **1**
Hierarchy check flag > **Check Hierarchy**
Element order > **High**
Number of Smoothing Iterations > **4**

Clear the screen and issue **HIDDEN,1;** (Display > DISPLAY OPTION > **Hidden Element Plot**) and **EPLOT** (Edit > PLOT > **Elements**) commands to obtain an element plot of the model without hidden lines. You can notice from the finite element mesh that at all three common boundaries, the elements do *not* match. The figure below shows a three dimensional view as well as 2D enlarged views of the mesh at the common boundaries.

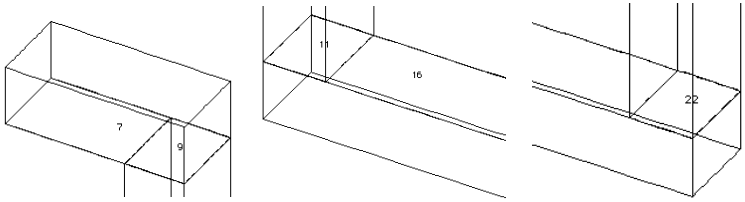
Figure 11-9. Generated Finite Element Mesh with Non-compatible Interfaces



Since the element types are incompatible, do ***not*** attempt to merge the nodes for this problem even if they appear to be coincident at the common boundaries. At interfaces with full compatibility, the nodes may however be merged.

You need to identify the surfaces that form the common boundary where the elements are mismatched. Clear the screen, activate surface label display using **ACTNUM,SF,1** (Control > ACTIVATE > **Entity Label**) command and plot all surfaces using **SFPLOT** (Edit > PLOT > **Surfaces**) command. You can zoom-in on the three interfaces for bond definition so that you can clearly identify the surface numbers for input. The figure below shows the enlarged views of the surfaces with labels at the common boundaries.

Figure 11-10. Surfaces at the Common Boundaries



From the above figure, you will find that:

- at the SOLID-to-SOLID element interface, surfaces 7 and 9 are interconnected and can be used for bond set 1,
- at the SOLID-to-TETRA10 element interface, surfaces 11 and 16 are interconnected, and can be used for bond set 2,
- at the TETRA10-to-TETRA10 element interface, surface 22 connects the two part entities and can be used for bond set 3.

The above information is helpful in bond definition using the command **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**). Select this command and specify the surface numbers for the first bond set as illustrated below:

Geo Panel: LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**
Bonding set > **1**
Primary Geometric entity type > **Surface**

Primary Surface > **9**
Secondary Geometric entity type > **Surface**
Beginning Surface > **7**
Ending Surface > **7**
Increment > **1**
Direction flag > **Bi Dir**

Repeat this procedure for the remaining two interfaces as shown below:

Geo Panel: LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**
Bonding set > **2**
Primary Geometric entity type > **Surface**

Primary Surface > **11**
Secondary Geometric entity type > **Surface**
Beginning Surface > **16**
Ending Surface > **16**
Increment > **1**
Direction flag > **Bi Dir**

Geo Panel: LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**
Bonding set > **3**
Primary Geometric entity type > **Surface**

Primary Surface > **22**
Secondary Geometric entity type > **Surface**
Beginning Surface > **22**
Ending Surface > **22**
Increment > **1**
Direction flag > **Bi Dir**

If you make a mistake in bond set definition, you can delete that set using the **BONDDEL** (LoadsBC > STRUCTURAL > BONDING > **Delete Bond Parameter**) command. Use the command **BONDLIST** (LoadsBC > STRUCTURAL > BONDING > **List**) to verify the geometric entities bonded together. This command will provide the bonding information between primary (or source) and secondary (or target) entities on the screen as shown below:

Set	Stype	Source	Ttype	#Targets	Targets
1	SF	9	SF	1	7
2	SF	11	SF	1	16
3	SF	22	SF	1	22

The actual constraint relations between the nodes of source and target geometric entities are formed and computed in the analysis stage. During this process, the bond sets defined above are further divided into subsets in which each source node is connected to one or more target nodes in its vicinity. It is possible to obtain a listing of the source and target nodes in the subsets, as explained a little later.




You can now proceed with applying boundary conditions and loads. Use the command **DSF** (LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Surface**) to restrain all degrees of freedom at the left end (surface 4) as illustrated below:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Surface**
Beginning Surface > **4**
Displacement Label > **ALL: All 6 DOF**
Value > **0.0**
Ending Surface > **4**
Increment > **1**

The pressure loading on the right end as well as the top surface of the right half is applied using the **PSF** (LoadsBC > STRUCTURAL > PRESSURE > **Define Surface**) command as illustrated below;

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define Surface**
Beginning Surface > **18**
Pressure Magnitude > **500**
Ending Surface > **18**
Increment > **1**
Pressure at the end of direction 1 > **500**
Pressure at the end of direction 2 > **500**
Pressure Direction > **Normal**

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define Surface**
Beginning Surface > **19**
Pressure Magnitude > **500**
Ending Surface > **19**
Increment > **1**
Pressure at the end of direction 1 > **500**
Pressure at the end of direction 2 > **500**
Pressure Direction > **Normal**

The input of material properties and element group data completes the model preparation phase:

Geo Panel: Propsets > **Pick Material Lib**
Material property set > **1**
Material Name > **A_STEEL**
Unit Label > **FPS**

Geo Panel: Propsets > **Element Group**
Element group > **1**
Element category > **Volume**
Element type (for volume) > **SOLID: 8- to 20-node 3D solid element**

Accept Defaults ...

Geo Panel: Propsets > **Element Group**
Element group > **2**
Element category > **Volume**
Element type (for volume) > **TETRA10: 10-node tetrahedral solid element**

Accept Defaults

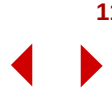
Before executing linear static stress analysis, you need to activate the input print flag using the command **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) in order to obtain a listing of primary node vs. secondary nodes bonding information.

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
Displacement Print Flag > **Yes**
... ...
Input Print Flag > **Yes**

As mentioned earlier, *do not* merge nodes for this example before performing analysis. Use the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) to execute a linear static stress analysis. The solution of this example requires about 30 Mb of free disk space on a personal computer with 16 Mb of memory.

After the analysis is successfully executed, you can inspect the output file (jobname.OUT) for results. You will notice that the bond information is written in the following form:



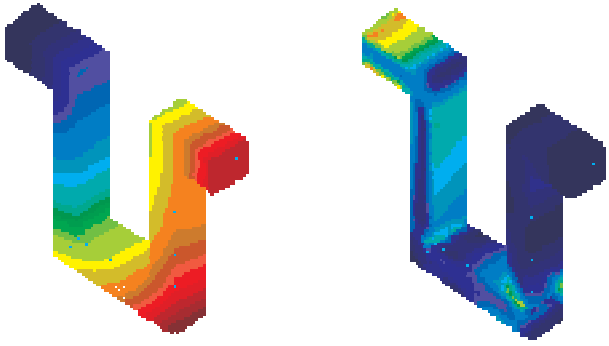



```
  Joint Bond coupling information
  -------------------------------

Ref.; Reference node
Types of coupling
   EQ. PTC ; Point to curve
   EQ. PTS ; Point to surface
Set Type  Ref.   NODE1 NODE2 NODE3 NODE4 NODE5 NODE6 NODE7 NODE8 NODE9 NODE10
  1  PTS  3224   3075  3079  3035  3031
  2  PTS  3225   3079  3083  3039  3035
  ... ...
```

The set in the above listing refers to the *subsets* derived from the bond sets you earlier defined using the **BONDDEF** (LoadsBC > STRUCTURAL > BONDING > **Define Bond Parameter**) command.

You can view the displacement and stress results after activating the required components using the (Results > PLOT > **Stress**, **Displacement**) command. The figures below show contours of resultant displacement and von Mises stresses. These figures reinforce the fact that the results obtained using the bond feature are continuous and smooth as if a fully compatible mesh were used.

Figure 11-11. Displacement and Stress



Linear Elastic Fracture Mechanics Analysis Examples

There are two quarter-point singular elements (8-node PLANE2D and 20-node SOLID) available in the Basic System to model problems with cracks. When you use these elements, the fracture parameter computed is the stress intensity factor for the applicable mode of fracture. Other fracture parameters such as the J-integral or the energy release (G) can be calculated based on the computed stress intensity factors or directly compute through the J_integral option of the Basic System. The following two examples illustrate the computation of stress intensity factors for problems with different crack geometry. It should be understood that in the following examples, due to structural symmetry, only half or one quarter of the structure is modeled. However, in more general cases, you may have to define a crack element for each one of the two faces of a crack.

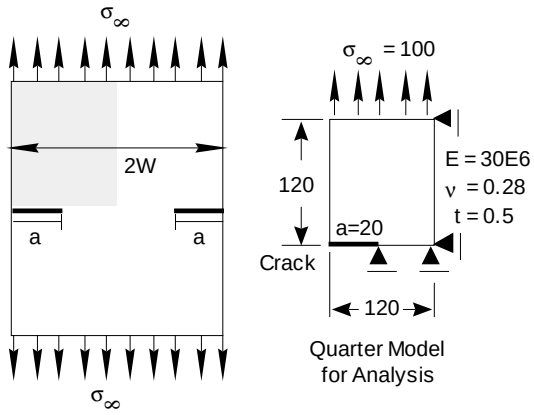
In the examples of this section only the use of crack element is considered in the evaluation of the fracture mechanics characteristics. However, you may use the J-integral option to evaluate the same for 2D problems (using TRIANG and PLANE2D elements), as well.

Example LEFM1 - Stress Intensity Factor of a Plate with Two Edge Cracks

The input for this problem is available to you in the file LEFM1.GEO. This example considers the mode-I stress intensity factor solution of a plate with two through-thickness edge cracks under far-field tensile stresses as shown in the figure below. The elastic material properties and other dimensions are shown in the figure below.




Figure 11-12. Problem for Stress Intensity Factor



The symmetry of the problem enables us to use a quarter of the model for analysis. Set the view to 2D X-Y using the Viewing (Binocular) icon. The geometry and the finite element mesh are created as illustrated in the following commands:

Geo Panel: Geometry > GRID > **Plane**
Rotation/Sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line style > **Solid**

Geo Panel: Display > VIEW PARAMETER > **View**
Accept defaults...

Geo Panel: Geometry > SURFACES > **Draw w/ 4 Coord**
Surface > **1**
XYZ-Coordinate of Keypoint 1 > **30,30,0**
XYZ-Coordinate of Keypoint 2 > **150,30,0**
XYZ-Coordinate of Keypoint 3 > **150,150,0**
XYZ-Coordinate of Keypoint 4 > **30,150,0**
Click on Auto Scale icon ...

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning Surface > **1**
Ending Surface > **1**
Increment > **1**

Number of nodes per element > **8**

Number of elements on the 1st curve > **12**

Number of elements on the 2nd curve > **15**

Spacing ratio for 1st curve > **1.0**
Spacing ratio for 2nd curve > **1.0**

Note that the mesh was generated with higher order elements in order to use the quarter-point element. The PLANE2D element required to be used for this problem is selected using the **EGROUP** (Propsets > **Element Group**) command whereas the material and sectional property data are entered as follows:

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Category > **Area**
Element Type (for area) > **PLANE2D: 2D 4- to 8-node plane element**

Accept Defaults ...

Geo Panel: Propsets > **Pick Material Lib**
Material Property set > **1**
Material Name > **A_STEEL**
Unit Label > **FPS**

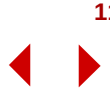
Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start loc. of the real const > **1**
No. of real const to be entered > **2**

RC1: Thickness > **0.5**
RC2: Material angle (Beta) > **0.0**

Symmetric boundary conditions are enforced along the vertical and horizontal axes of symmetry. Since the crack represents a traction free edge and a discontinuity in the geometry, there are no displacement boundary conditions enforced on the crack. Therefore, after you apply displacement boundary conditions on the curve entity, you need to remove the constraints placed on the crack. The following commands illustrate:



Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
Beginning curve > **1**
Displacement Label > **UY: Y translation**
Value > **0.0**
Ending curve >
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
Beginning curve > **4**
Displacement Label > **UX: X translation**
Value > **0.0**
Ending curve > **4**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Delete Nodes**
Beginning node > **1**
Displacement Label > **UY: Y translation**
Ending node > **4**
Increment > **1**

The far-field tensile stresses on the top edge can be applied using the **PCR** (LoadsBC > STRUCTURAL > PRESSURE > **Define Curves**) command as shown below:

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define Curves**
Beginning curve > **2**
Pressure magnitude > **-100**
Ending curve > **2**
Increment > **1**
Pressure at the end of direction 1 > **-100**
Pressure Direction > **Normal Direction**

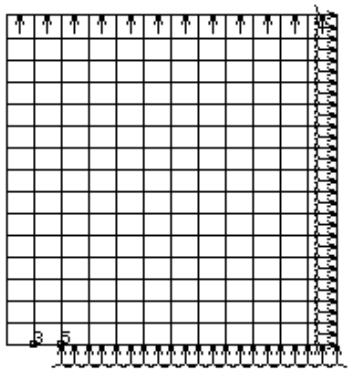



You can now proceed to define the quarter-point element at the crack tip. As outlined in Chapter 8, *Modeling Guidelines*, the command **CRACK** (Analysis > STATIC > CRACK > **Define Crack**) is used to identify the nodes at the crack tip. The first node is *always* at the crack tip, and the second node at the *corner* of the *same* element, away from the crack tip. The third node is required for 3D crack problems only. The **CRACK** (Analysis > STATIC > CRACK > **Define Crack**) command is illustrated below:

Geo Panel: Analysis > STATIC > CRACK > **Define Crack**
Crack number > **1**
Node 1 > **5**
Node 2 > **3**
Node 3 >

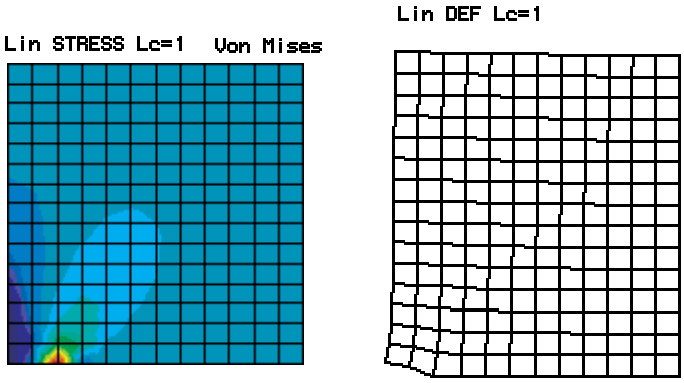
The commands **CRACKLIST** and **CRACKDEL** (Analysis > STATIC > CRACK > **List Crack, Delete Crack**) from the CRACK submenu can be used to list or delete quarter-point elements defined. The figure below shows the finite element model with applied loading and boundary conditions. The figure also shows the node numbers at the crack tip and the corner of the quarter-point element.

Figure 11-13. Finite Element Model with Boundary Conditions and Load



You can now proceed to performing a linear static stress analysis of the model using the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**). The Basic System computes the stress intensity factors and prints them in the output file. To see the effect of geometric singularity introduced by the crack on the stress distribution, you can use the command **STRPLOT** (Results > PLOT > **Stress**). The figures below show a plot of von Mises stresses as well as the deflected shape of the model.

Figure 11-14. Stress Contours and Deformed Shape Plot of the Cracked Plate



The stress intensity factors are read from the output file (jobname.OUT). For this example, they were found to be:

```
     CRACK STRESS INTENSITY FACTORS FOR CRACK 1

       KI    =  0.885133E+03
```

Note that the mode-II stress intensity factor is not relevant for this problem. The analytical solution for the mode-I stress intensity factor is as follows (Keer and Freedman, 1973):

$$K_I = \sqrt{\pi a}\,\frac{1.12 - 0.61\frac{a}{W} + 0.13\frac{a^3}{W^3}}{\sqrt{1-\frac{a}{W}}} \quad \text{for all a/W} \qquad (11\text{-}2)$$

The table below shows a comparison between the analytical and COSMOS/M solutions.

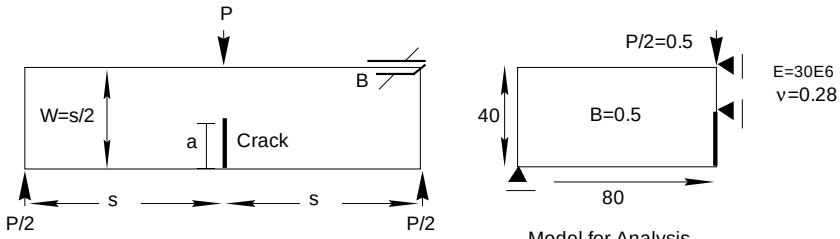
Table 11-1. Comparison Between COSMOS/M and Analytical Solution for KI

KI Solution	Result	Error (%)
Theoretical	884.763	N/A
COSMOS/M 300 elements	885.133	0.42

Example LFEM2 - Stress Intensity Factor of Three Point Bend Specimen

The input for this problem is available to you in the file LEFM2.GEO. This example considers the mode-I stress intensity factor solution of a beam with a through-thickness edge crack at the center subjected to a point load as shown in the figure below. Otherwise known as a 3-point bend specimen, this type of model is commonly used to experimentally determine the mode-I stress intensity factor. The elastic material properties and other dimensions are shown in the figure below.

Figure 11-15. Problem for Stress Intensity Factor Computation



The symmetry of the problem enables us to use one half of the model for analysis. Change the view to 2D X-Y using the View (Binocular) icon. The geometry and the finite element mesh are created as illustrated in the following commands:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line Style > **Solid**

Geo Panel: Geometry > SURFACES > **Draw w/ 4 Coord**
Surface > **1**
XYZ-Coordinate of Keypoint 1 > **20,20,0**
XYZ-Coordinate of Keypoint 2 > **100,20,0**
XYZ-Coordinate of Keypoint 3 > **100,60,0**
XYZ-Coordinate of Keypoint 4 > **20,60,0**
Click on Auto Scale icon ...

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **1**
Ending surface > **1**
Increment > **1**
Number of nodes per element > **8**

Number of elements on the 1st curve > **20**
Number of elements on the 2nd curve > **10**

Accept Defaults...

As in the previous example, the mesh was generated with higher order elements in order to use the quarter-point element. The PLANE2D element required to be used for this problem and the material and sectional property data are entered as follows:

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Category > **Area**
Element Type (for area) > **PLANE2D: 2D 4- to 8-node plane element**

Accept Defaults ...

Geo Panel: Propsets > **Pick Material Lib**
Material Property set > **1**
Material Name > **A_STEEL**
Unit Label > **FPS**

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real Constant set > **1**

Start loc. of the real const > **1**
No. of real const to be entered > **2**

RC1: Thickness > **0.5**
RC2: Material angle (Beta) > **0.0**

Symmetric boundary conditions are enforced along the vertical axis of symmetry. Since the crack represents a free edge, there are no displacement boundary conditions enforced on the crack. Along the vertical axis of symmetry (except at the crack), the horizontal displacements are specified to be zero. The boundary conditions are input as shown below:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
Beginning node > **1**
Displacement Label > **UY: Y translation**
Value > **0.0**
Ending node > **1**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
Beginning node > **351**
Displacement Label > **UX: X translation**
Value > **0.0**
Ending node > **351**
Increment > **1**

Repeat the above command for UX at nodes 372, 413, 434, 475, 496, 537, 558, 599, 620 and 661. The point load at the top is specified as shown below:

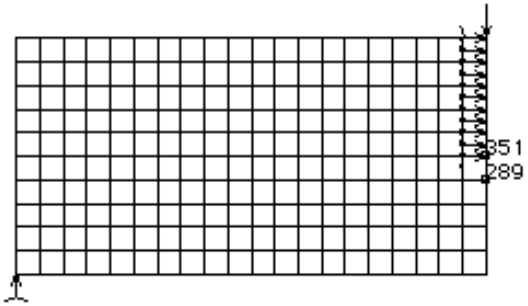
Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Node**
Beginning node > **661**
Force label > **FY: Y force**
Value > **-0.5**
Ending node > **661**
Increment > **1**

You can now proceed to define the quarter-point element at the crack tip. For this example, the first node which is *always* at the crack tip is 351, and the second node which is at the *corner* of the *same* element away from the crack tip, is 289. These nodes are specified using the **CRACK** (Analysis > STATIC > CRACK > **Define Crack**) command as illustrated below:

Geo Panel: Analysis > STATIC > CRACK > **Define Crack**
Crack number > **1**
Node 1 > **351**
Node 2 > **289**
Node 3 > **289**

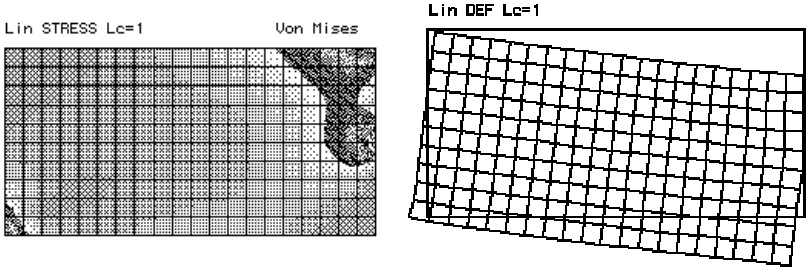
The figure below shows the finite element model with loads and boundary conditions. The figure also shows the node numbers at the crack tip and the corner of the quarter-point element.

Figure 11-16. Finite Element Model with Boundary Conditions and Loads



You can now proceed to performing a linear static stress analysis of the model using the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**). The effect of geometric singularity introduced by the crack on the stresses can be viewed using the command **STRPLOT** (Results > PLOT > **Stress**). The figures below show a plot of von Mises stresses as well as the deflected shape of the model.

Figure 11-17. Stress Contours and Deformed Shape Plot of the Cracked Plate



The stress intensity factors are read from the output file (jobname.OUT). For this example, they were found to be:

```
CRACK STRESS INTENSITY FACTORS

KI    =  0.294492E+01
```

Once again, the mode-II stress intensity factor is not relevant for this problem. The analytical solution for the mode-I stress intensity factor is as follows (Gross and Srawley, 1972):

$$K_I = 3.75 \frac{PW}{B(W-a)^{\frac{3}{2}}} \qquad for \quad 0.4 \le a/W \; 0.6 \tag{11-3}$$

The table below shows a comparison between the analytical and Basic System solutions. A second finite element solution obtained with 800 elements, shows convergence towards the analytical solution for mode-I stress intensity factor.

Table 11-2. Comparison Between COSMOS/M and Analytical Solution For K_I

KI Solution	Result	Error (%)
Theoretical	3.3541	N/A
COSMOS/M 200 elements	2.94492	12.2
COSMOS/M 800 elements	3.03311	9.57

Buckling Analysis Examples

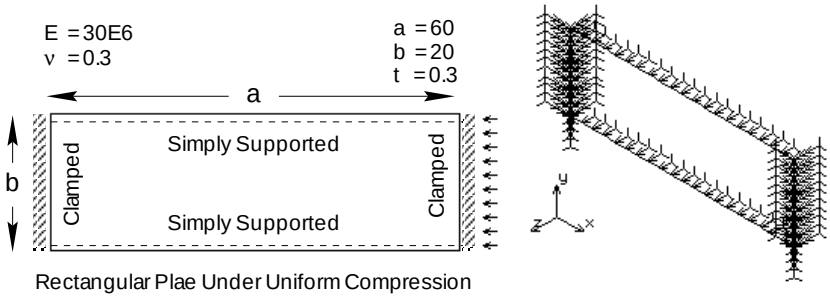
Buckling analysis which determines the critical buckling loads and mode shapes can be performed on many types of practical problems in the Basic System. Various eigenvalue extraction methods and modeling features available in the DSTAR module will enable you to address almost all types of modeling and analysis situations. Owing to space limitations, it is not possible to demonstrate all of these features. The example presented here discusses the application of some of the features outlined in Chapter 1. In addition, the second part of this manual presents more than twelve verification problems you may find useful in understanding many aspects of the buckling analysis capabilities available.

✍ The second part of this manual presents more than 100 verification problems on all analysis features of the Basic System. You are recommended to use these examples to learn more about the Basic System.

Example BUCKL1 - Buckling of a Rectangular Plate Under Uniform Pressure

The input for this example is available in the file BUCKL1.GEO. This example considers the eigenvalue buckling of a rectangular plate under uniform pressure. The material properties and dimensions of the plate are shown in the figure below. Two opposite edges of the plate are simply supported whereas the remaining two edges are clamped. The plate is subjected to a uniform pressure loading of unit intensity so that the computed eigenvalues will directly yield the critical buckling loads. Otherwise, the eigenvalues are treated as multipliers to the actual buckling loads.

Figure 11-18. Problem for Critical Buckling Load and Applied Boundary Conditions



One of the clamped edges of the plate is subjected to uniform pressure. However, in order to induce buckling in the finite element model, the x-component of the displacements was *released* on this edge whereas the other clamped edge was restrained with respect to all components of displacements. The figure above also shows the applied boundary conditions on the plate in 3D space.

The geometry and the finite element mesh of the plate are created as illustrated below:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line Style > **Solid**

Geo Panel: Geometry > SURFACES > **Draw w/ 4 Coord**
Surface > **1**
XYZ-Coordinate of Keypoint 1 > **20,20,0**
XYZ-Coordinate of Keypoint 2 > **80,20,0**
XYZ-Coordinate of Keypoint 3 > **80,40,0**
XYZ-Coordinate of Keypoint 4 > **20,40,0**

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **1**
Ending surface > **1**
Increment > **1**
Number of nodes per element > **4**

Number of elements on 1st curve > **20**

Number of elements on 2nd curve > **10**
Accept Defaults

Click on Auto Scaling icon to view the model clearly. The simply supported boundary condition on two edges of the plate was modeled by restraining the z-component of displacements. The pressure loading and the displacement boundary conditions are applied as shown in the commands below:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
Beginning curve > **1**
Displacement label > **UZ: Z translation**
Value > **0.0**
Ending curve > **4**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
Beginning curve > **3**
Displacement label > **ALL: All 6 DOF**
Value > **0.0**
Ending curve > **3**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
Beginning curve > **4**
Displacement label > **UY: Y translation**
Value > **0.0**
Ending curve > **4**
Increment > **1**

Displacement label > **RX: X rotation**
Displacement label > **RY: Y rotation**
Displacement label > **RZ: Z rotation**

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define Curves**
Beginning curve > **4**
Pressure magnitude > **1**
Ending curve > **4**
Increment > **1**
Pressure at the end of direction 1 > **1**
Pressure direction > **Normal Direction**

The plate will be modeled with SHELL4 elements selected using the **EGROUP** (Propsets > **Element Group**) command. The material and sectional property data

are entered using **MPROP** (Propsets > **Material Property**) and **RCONST** (Propsets > **Real Constant**) commands as follows:

Geo Panel: Propsets > **Element Group**
Element group > **1**
Element category > **Area**
Element type (for area) > **SHELL4: 4-node thin shell element**

OP1: Type > **QUAD4**
... ...
Accept Defaults

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material property name > **EX: Elasticity modulus 1st dir**
Property value > **30E6**

Material property name > **NUXY: Poisson ratio 2nd to 1st**
Property value > **0.3**

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real constant set > **1**

Start loc. of the real const. > **1**
No. of real const. to be entered > **6**
RC1: Thickness > **0.3**
... ...
Accept Defaults

The number of buckling modes as well as the method of eigenvalue extraction is specified using the **A_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Buckling Options**) command. We will input for the first five modes of buckling using Lanczos method for this problem as shown below:

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Buckling Options**
Number of Eigenvalues > **5**
Method > **Lanczos method**
... ...
Accept Defaults

The command **R_BUCKLING** (Analysis > FREQUENCY/BUCKLING > **Run Buckling**) can now be issued to compute the critical buckling loads. The results of execution are written to the output file, jobname.OUT, and you can examine this file for buckling load factors and the corresponding mode shape deflections.

The critical buckling loads can also be listed on the screen using the **FREQLIST** (Results > LIST > **Natural Frequency**) command. The listing below shows the buckling load factors for the first five modes:

```
   B U C K L I N G    E I G E N V A L U E (S)
                  by
   L A N C Z O S    A L G O R I T H M

     EIGENVALUE       EIGENVALUE
       NUMBER

          1         0.2712670E+05
          2         0.2775493E+05
          3         0.3398611E+05
          4         0.3840657E+05
          5         0.5011349E+05
```

The analytical solution for this problem can be found in Roark's Formulas for Stress and Strain (Sixth Edition, Young, 1989) and is given below for a/b = 3:

$$\sigma_c = 3.63 \frac{E}{1-\nu^2}\left(\frac{t}{b}\right)^2 \qquad (11\text{-}4)$$

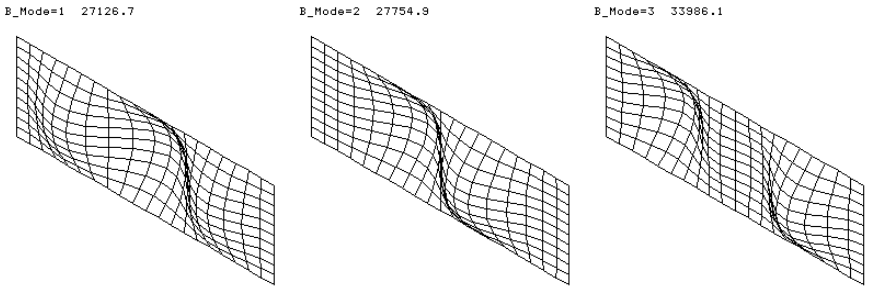
where *b* is the length of the shorter side, and *t* is the thickness. The following table shows the comparison between the analytical and the Basic System solutions.

Table 11-3. Comparison Between COSMOS/M and Analytical Solution

First Buckling Mode Solution	Result	Error (%)
Analytical	26925.82	N/A
COSMOS/M 200 elements	27126.70	0.75

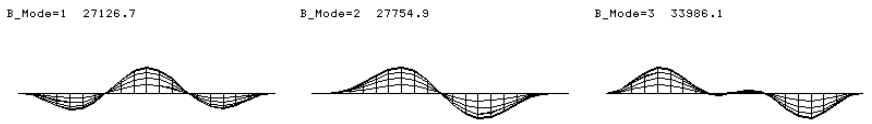
As seen in the above table, the Basic System solutions are very accurate. The command **DEFPLOT** (Results > PLOT > **Deformed Shape**) can be used to plot the buckling mode shapes. The first three buckling modes are shown below in the isometric viewing position.

Figure 11-19. Buckling Mode Shapes of the Plate In 3-D Space



You can change the viewing position to 2D space to see the lateral deformations clearly. The figure below shows the three buckling modes corresponding to the above figure in the side view position.

Figure 11-20. Corresponding Buckling Mode Shapes In 2D Space



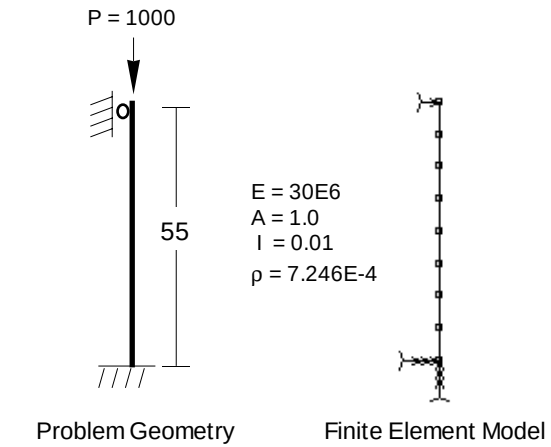
Modal Analysis Examples

Modal analysis which determines the natural frequencies and mode shapes can be performed on many types of practical problems in the Basic System. Various eigenvalue extraction methods and modeling features available in the DSTAR module will enable you to address almost all types of modeling and analysis situations. Owing to space limitations, it is not possible to demonstrate all of these features. Two examples presented here (MODAL1 and MODAL2) discuss the application of some of the features outlined in Chapter 1. In addition, the second part of this manual presents many verification problems you may find useful in understanding many aspects of the modeling and analysis capabilities.

Example MODAL1 - In-plane Effects on the Natural Frequencies of a Column

The input for this problem is available to you in file MODAL1.GEO. For slender structures with in-plane loading, the natural frequencies are significantly altered depending on the type of preload applied. As explained in Chapter 2, *Mathematical Background*, compressive loads tend to decrease the natural frequencies whereas tensile preloads increase them. The effect of a preload on the natural frequencies is significant for the first few modes and its influence gradually diminishes for higher modes. This example quantitatively demonstrates the effects of compressive and tensile preloads on the natural frequencies with respect to those obtained with no preloads.

Figure 11-21. Problem for Modal Analysis with In-Plane Effects



The geometry and the finite element mesh for this problem are created as illustrated in the following commands:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
... ...
Accept Defaults

Geo Panel: Geometry > CURVES > **Draw Polyline**
Curve > **1**
XYZ-Coordinate of Keypoint 1 > **50,0,0**
XYZ-Coordinate of Keypoint 2 > **50,55,0**
XYZ-Coordinate of Keypoint 3 > **50,55,0**

Geo Panel: Meshing > PARAMETRIC MESH > **Curves**
Beginning curve > **1**
Ending curve > **1**
Increment > **1**
Number of nodes per element > **2**

Number of elements on each curve > **8**
Spacing ratio > **1.0**

The column will be modeled using 2D beam elements (BEAM2D). This element is selected using the **EGROUP** (Propsets > **Element Group**) command whereas the material and sectional property data are entered as follows:

Geo Panel: Propsets > **Element Group**
Element group > **1**
Element category > **Line**
Element type (for line) > **BEAM2D: 2D elastic beam element**

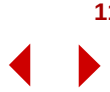
Accept Defaults

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real constant set > **1**

Start loc. of the real const. > **1**
No. of real const. to be entered > **8**

RC1: Cross-sectional area > **1.0**
RC2: Moment of inertia IZ > **0.01**
RC3: Depth > **1.0**



Geo Panel: Propsets > **Pick Material Lib**
Material Property set > **1**
Material name > **A_STEEL**
Unit label > **FPS**

The displacement boundary conditions at the top and bottom are enforced using the **DND** (LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**) command where as the vertical load on the column is specified using the **FND** (LoadsBC > STRUCTURAL > FORCE > **Define Nodes**) command. *In the first run, we will obtain natural frequencies in the presence of a compressive preload*. The following lines illustrate the use of these commands:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
Beginning node > **1**
Displacement label > **ALL: All 6 DOF**
Value > **0.0**
Ending node > **1**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
Beginning node > **9**
Displacement label > **UX: X translation**
Value > **0.0**
Ending node > **9**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **9**
Force label > **FY: Y force**
Value > **-1000**
Ending node > **9**
Increment > **1**

The effect of a preload on the natural frequencies is specified by activating the in-plane effects flag in the **A_FREQUENCY** (Analysis > FREQUENCY/ BUCKLING > **Frequency Options**) command as illustrated below. The number of eigenvalues are also input in the same command. We will investigate the first five modes of the column.

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Frequency Options**
Number of frequencies > **5**
Method > **Subspace iteration method**
Maximum number of iterations > **16**
Sturm sequence flag > **No**
Shift flag > **No Eigenvalue Shift**
Shift value > **0.0**
In-plane effect flag > **Yes**
... ...
Accept Defaults

You can use the command **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) to compute the first five natural frequencies of the column in the presence of a compressive preload. After the analysis is successfully executed, you can use the command **FREQLIST** (Results > LIST > **Natural Frequency**) to list the computed natural frequencies on the screen.

Delete the load at the top of the column and define a *tensile* preload as illustrated below:

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Delete Nodes**
Beginning node > **9**
Force label > **FY: Y force**
Ending node > **9**
Increment > **1**

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **9**
Force label > **FY: Y force**
Value > **1000**
Ending node > **9**
Increment > **1**

Before executing another normal modes analysis, you need to activate a flag so that the results of the analysis you last performed are *not* over-written with the new results. Use the command **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) as illustrated below:




Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
Displacement Print Flag > **Yes**
... ...
Output Flag > **Append**

Re-execute the problem using the command **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) to compute the first five natural frequencies of the column in the presence of a *tensile* preload.

Delete the load at the top of the column using the command **FNDEL** (LoadsBC > STRUCTURAL > FORCES > **Delete Nodes**) and deactivate the in-plane effects flag in the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command. Execute another analysis to obtain results *without* the effects of preloads. Again, you can use the command **FREQLIST** (Results > LIST > **Natural Frequency**) to list the computed natural frequencies. The results are also written to the output file (jobname.OUT).

The table below shows a summary of results for the first five modes. The results clearly show the effects of tensile and compressive preloads on the natural frequencies. As stated earlier, the influence of preloads on the natural frequencies decrease for higher mode shapes.

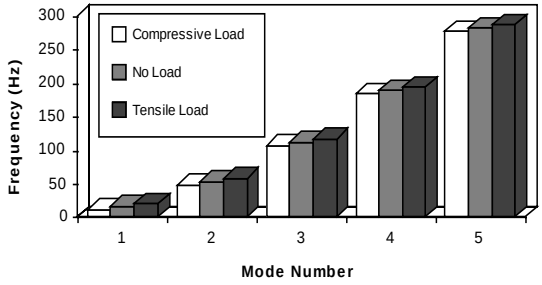
Table 11-4. Comparison of Natural Frequencies With and Without Preloads

Mode Number	In-plane Load - Frequency (Hz)		
	Compressive	No	Tensile
1	11.71	16.44	20.03
2	48.43	53.26	57.68
3	105.92	110.83	115.54
4	183.02	187.90	192.66
5	275.20	279.88	284.47

The effect of tensile and compressive preloads on the natural frequencies are better represented in the following figure.

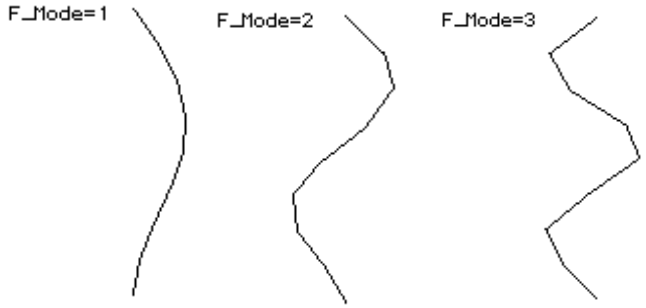



Figure 11-22. Effect of Tensile and Compressive Preloads on the Natural Frequencies



The **DEFPLOT** (Results > PLOT > **Deformed Shape**) command can be used for plotting the free vibration modes of the column. The figure below shows the first three modes for the column with no preloads.

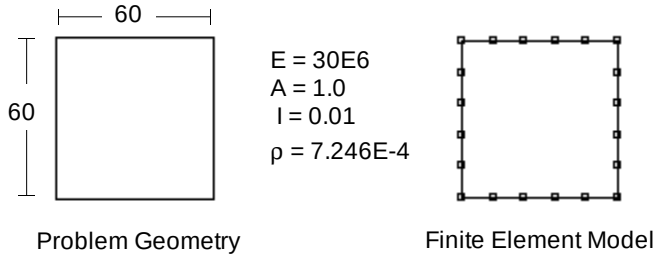
Figure 11-23. Free Vibration Modes of the Column



Example MODAL2 - Modal Analysis of a Square Frame with Rigid Body Modes

The input for this problem is available to you in file MODAL2.GEO. The analysis problem consists of a box-shaped frame with material and cross sectional properties and dimensions as shown in the figure below. The frame which lies in 2D space is unsupported, and it is required to determine the first ten natural frequencies and mode shapes.

Figure 11-24. Problem for Modal Analysis with Rigid Body Modes



The geometry and the finite element mesh for this problem are created as illustrated in the following commands:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line Style > **Solid**

Geo Panel: Geometry > CURVES > **Draw Polyline**
Curve > **1**
XYZ-Coordinate of Keypoint 1 > **20,20,0**
XYZ-Coordinate of Keypoint 2 > **80,20,0**
XYZ-Coordinate of Keypoint 3 > **80,80,0**
XYZ-Coordinate of Keypoint 4 > **20,80,0**
XYZ-Coordinate of Keypoint 5 > **20,20,0**

Geo Panel: Meshing > PARAMETRIC MESH > **Curves**
Beginning curve > **1**
Ending curve > **4**
Increment > **1**
Number of nodes per element > **2**

Number of elements on each curve > **5**
Spacing ratio > **1.0**

Click on the View (Binocular) icon and change the view to X-Y. You can scale the image to fit the viewport by clicking on the Auto Scale icon. The frame will be modeled with 2D beam elements (BEAM2D). This element is selected using the **EGROUP** (Propsets > **Element Group**) command whereas the material and sectional property data are entered as follows:

Geo Panel: Propsets > **Element Group**
Element group > **1**
Egroup category > **Line**
Element type (for line) > **BEAM2D: 2D elastic beam element**

Accept Defaults

Geo Panel: Propsets > **Real Constant**
Associated Element group > **1**
Real constant set > **1**

Start loc. of the real const. > **1**
No. of real const. to be entered > **8**

RC1: Cross-sectional area > **1**
RC2: Moment of inertia (IZ) > **0.01**
RC3: Depth > **1**
... ...
Accept Defaults

Geo Panel: Propsets > **Pick Material Lib**
Material Property set > **1**
Material name > **A_STEEL**
Unit label > **FPS**

Since the finite element mesh for each curve is generated independently, there will be two nodes at the corners. For compatibility requirements, these nodes have to be merged into one. The **NMERGE** (Meshing > NODES > **Merge**) and **NCOMPRESS** (Edit > COMPRESS > **Nodes**) commands are used to merge the nodes and consecutively number them, respectively.




Since the behavior of the finite model is confined to 2D space, you can expect to see three rigid body modes: two translations and a rotation. As described in Chapter 2, *Mathematical Background*, problems with rigid body modes can be dealt with in two ways: using an eigenvalue shift, or a soft spring addition, specified using the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command. We will use both options and compare the results.

We will compute the first ten free vibration modes and mode shapes for this problem. The number of eigenvalues as well as the eigenvalue shift are input in the **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Frequency Options**) command as illustrated below:

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Frequency Options**
Number of frequencies > **10**
Method > **S: Subspace iteration**
Maximum number of iterations > **16**
Sturm sequence flag > **No**
Shift flag > **Shift by program**
Shift value > **0.0**
... ...
Accept Defaults

The command **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) is next executed to perform modal analysis of the square frame. After the analysis is successfully executed, you can use the command **FREQLIST** (Results > LIST > **Natural Frequency**) to list the computed natural frequencies on the screen. The table below shows the mode numbers and the corresponding natural frequencies using the shift option. The first three modes shown represent rigid body modes with negligible frequencies. This table also shows the frequencies obtained using the soft spring option, explained a little later.




Table 11-5. Comparison of Natural Frequencies Using Eigenvalue Shift and Soft Spring Option

Mode Number	Option Frequency (Hz)	
	Shift	Soft Spring
1	2.40086e-06	1.70049e-03
2	9.05758e-06	1.70051e-03
3	1.25760e-05	1.70122e-03
4	5.08984e+00	5.08985e+00
5	8.84410e+00	8.84410e+00
6	1.71626e+01	1.71626e+01
7	1.71626e+01	1.71626e+01
8	2.00285e+01	2.00285e+01
9	3.74398e+01	3.74398e+01
10	4.47586e+01	4.47586+01

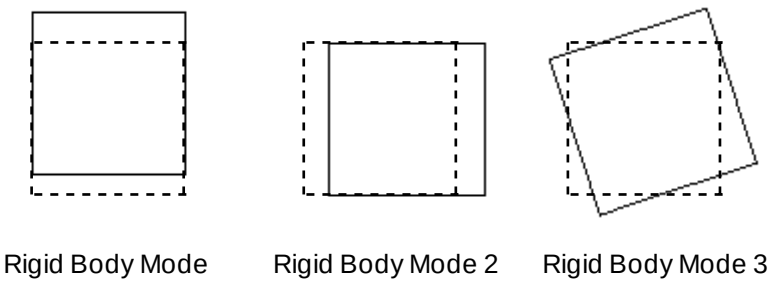
The command **DEFPLOT** (Results > PLOT > **Deformed Shape**) can be used to plot the mode shapes. The rigid body modes can also be computed using the soft spring option. As explained in Chapter 2, *Mathematical Background*, the addition of soft spring stiffness results in each diagonal term of the structural stiffness incremented by a small value to provide numerical stability. In the Basic System, you can vary the value of soft spring stiffness. The soft spring addition is specified as shown below:

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Frequency Options**
Number of frequencies > **10**
Method > **S: Subspace iteration**
Maximum number of iterations > **16**
Sturm sequence flag > **No**
Shift flag > **No Eigenvalue Shift**
Shift value > **0.0**
In-plane effect > **No**
Tolerance > **0.000010**
Soft Spring flag > **Yes**
... ...
Accept Defaults

Use the command **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) to append the new analysis results to the previous one in the output file. Next, use the command **R_FREQUENCY** (Analysis > FREQUENCY/BUCKLING > **Run Frequency**) to compute the natural frequencies and mode

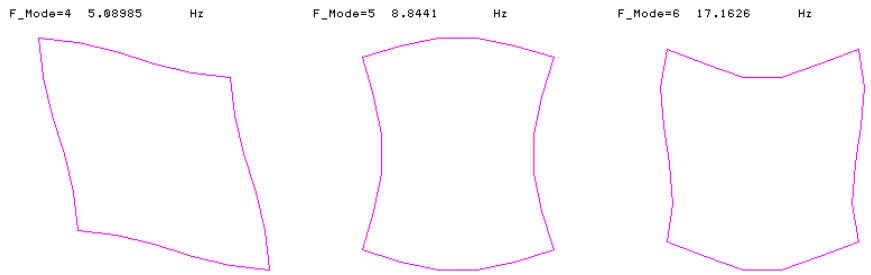
shapes of the square frame. After the analysis is successfully executed, you can use the command **FREQLIST** (Results > LIST > **Natural Frequency**) to list the computed natural frequencies on the screen. As shown in the previous table, the first three modes representing rigid body displacements have negligible natural frequencies. You can notice from the table that the natural frequencies for the deformable modes (mode 4 onwards) are *identical* with those obtained using the eigenvalue shift option. The figure below shows the three rigid body modes obtained using the soft spring option.

Figure 11-25. Rigid Body Modes Computed Using Soft Spring Option



The **DEFPLOT** (Results > PLOT > **Deformed Shape**) command can be used in either approach for plotting the deformable free vibration modes of the square frame. The figure below shows the fourth, fifth and six free vibration modes.

Figure 11-26. Deformable Free Vibration Modes of the Square Frame Using Soft Spring Option



Examples on Submodeling

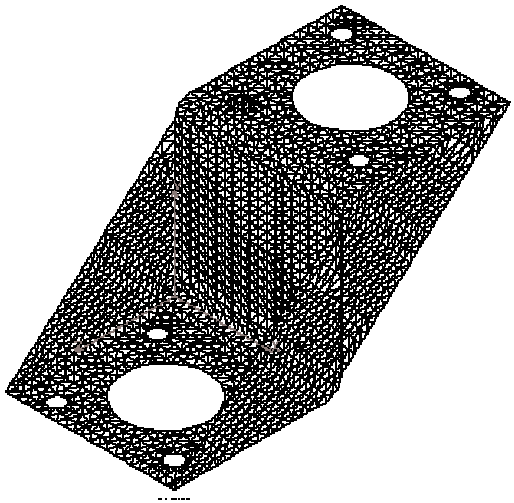
Submodeling is a feature of STAR which can be used to improve the solution at locally critical areas without having to rerun for the whole model. This can reduce the CPU time substantially for large problems.

Example LSUBM1 - Using Submodeling for Shell Problems

The input for the modeling part of this example is available in the file LSUBM1.GEO. The figure below shows a 3D view of the model constrained at one end and subjected to forces at the other end.

After the initial run of the model the von Mises stress distribution viewed from a more convenient orientation, is displayed in the following figure. Change the view to X-Z using the Viewing icon.

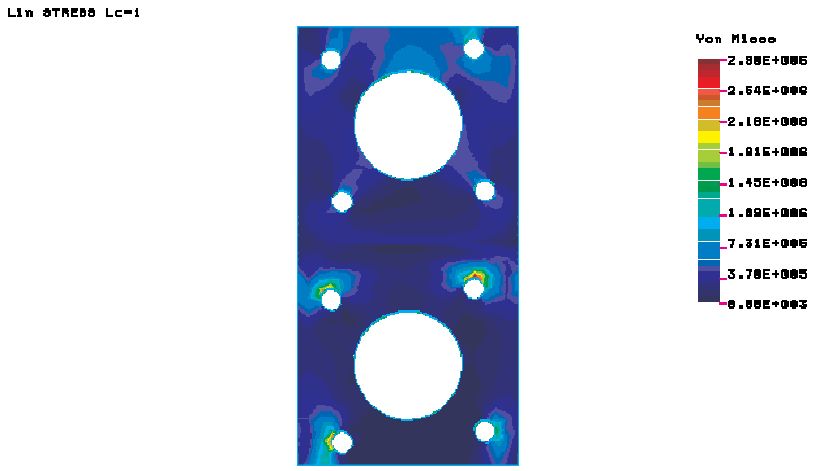
Figure 11-27. Model Geometry for the Analysis



Geo Panel: Results > PLOT > **Stress**
Load case number > **1**
Component > **VON**
Layer number > **1**
Coordinate system > **0**
Stress Flag > **Nodal Stress**
Face of element > **Top Face**

Select Contour and Accept Defaults

Figure 11-28. von Mises Stress Distribution of the Original Run



To improve the results at the areas of high stress concentrations, four portions surrounding the constrained holes are selected by repeated use of **SELWIN** (Control > SELECT > **by Windowing**) command (see the LSUBM1.GEO file for selected elements):

After the element selection is made to define the submodel, use the **SUBMODEL** (Analysis > STATIC > **Define SubModel**) command to activate submodeling and refine the local mesh. Run STAR to solve for the submodel.

Geo Panel: Analysis > STATIC > **Define SubModel**
Submodeling option > **On**

Push boundary node flag > **No**

Geo Panel: Analysis > STATIC > **Run Static Analysis**

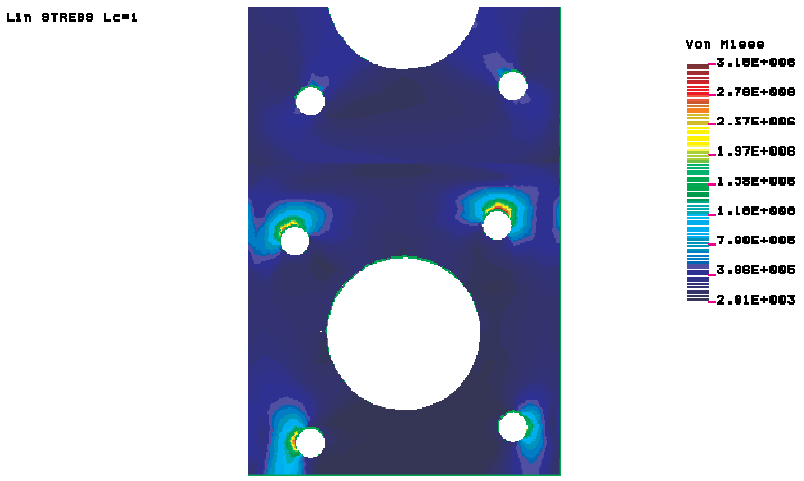
The following figure displays von Mises stress results. Note that you need to bring back the rest of the model (not analyzed with the submodel) to visualize results on the entire model.

Geo Panel: Control > SELECT > **Initialize**
Entity name > **Elements**
... ...
Accept Defaults

Geo Panel: Results > PLOT > **Stress**
Load case number > **1**
Component > **VON**
Layer number > **1**
Coordinate system > **0**
Stress Flag > **Nodal Stress**
Face of element > **Top Face**

Select Contour and Accept Defaults

Figure 11-29. von Mises Stress Distribution After Running the Submodel



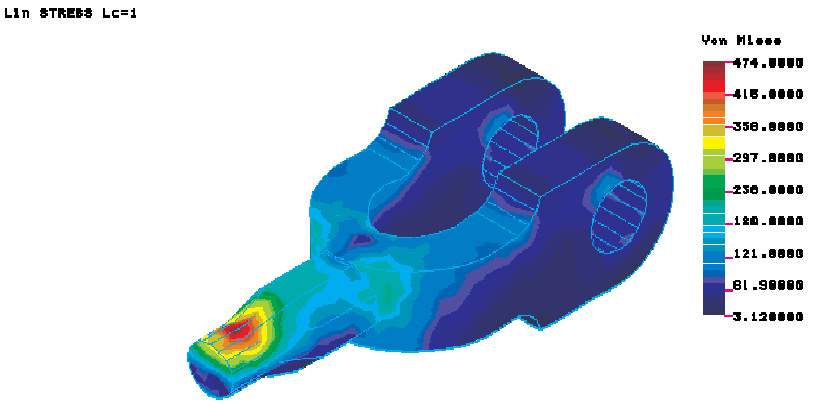
The maximum von Mises stresses calculated by the submodel shows a modification of about 10% over the original run (diagram).

If you deactivate submodeling (use the **SUBMODEL** (Analysis > STATIC > **Define SubModel**) command to deactivate submodeling) and run the program for the entire model it takes more than 20 minutes to complete the job on a 66 Hz, 486 PC, whereas it takes only 3 minutes to run for the submodel. Time savings increase substantially as the problem size becomes larger. More interestingly, the maximum von Mises stress obtained in the entire model run (including the refined mesh) is almost the same as the one calculated by the submodel. The accuracy obtained by the submodel depends on how far are the boundaries of the submodel from the stress concentration areas.

Example LSUBM2 - Using Submodeling with Tetrahedral Elements

The input for the model of this example is available in the file LSUBM2.GEO. The figure below shows von Mises stress distribution of the joint after the initial run.

Figure 11-30. von Mises Stresses for the Initial Run



A view along the Y-direction (X-Z plane) is chosen for a better selection of the submodel region, using the Viewing icon.

Geo Panel: Control > SELECT > **by Windowing**
... Select Desired Area ...

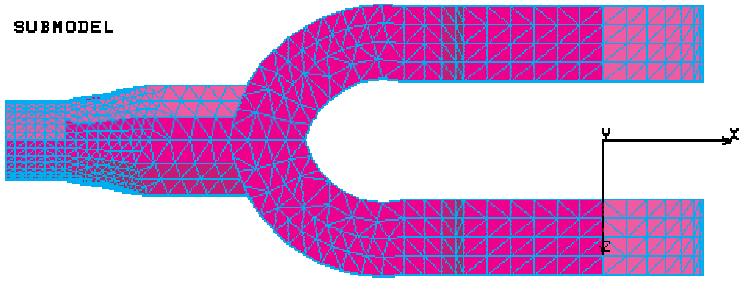
(Note: alternatively, you may use the **SELRANGE** (Control > SELECT > **by Range**) command to select partially through the depth of the model).

Geo Panel: Analysis > STATIC > **Define SubModel**
Submodeling option > **On**

Push boundary node flag > **No**

Geo Panel: Analysis > STATIC > **Run Static Analysis**

Figure 11-31. Region Selected for Submodeling



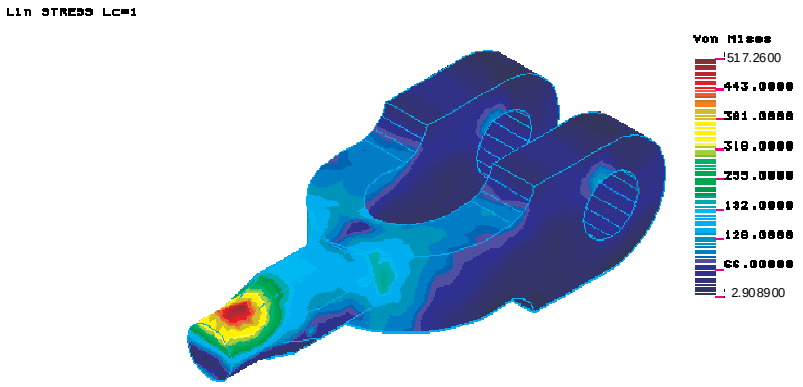
The following figure displays the von Mises stresses after running for the submodel. You can change the view to isometric by using the Viewing icon.

Geo Panel: Control > SELECT > **Initialize**
Entity name > **Elements**
... ...
Accept Defaults

Geo Panel: Results > PLOT > **Stress**
Load case number > **1**
Component > **VON**
Layer number > **1**
Coordinate system > **0**
Stress Flag > **Nodal Stress**
Face of element > **Top Face**

Select Contour and Accept Defaults

Figure 11-32. von Mises Stresses for the Submodel

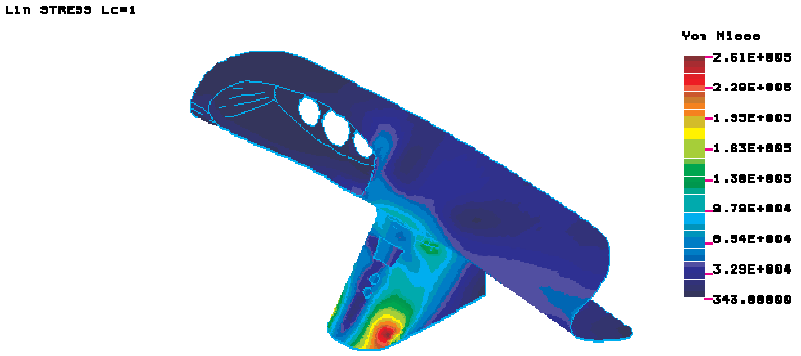


Maximum von Mises stresses are increased form the initial value of 470 psi to 517 psi after solving for the submodel.

Example LSUBM3 - Using Submodeling for Shell Problems

The input for this problem is provided to you in the file LSUBM3.GEO. The model represents a car dashboard subjected to surface pressure and concentrated loads. The figure below shows von Mises stresses after the initial run. To improve the results at the lower part of the model, the **SELWIN** (Control > SELECT > **by Windowing**) command with the circular window option is used to select a circular area to define the submodel.

Figure 11-33. von Mises Stress for the Initial Run



Geo Panel: Analysis > STATIC > **Run Static Analysis**

(For initial run ...)

Geo Panel: Control > SELECT > **by Windowing**

(Select critical area for submodeling)

Geo Panel: Analysis > STATIC > **Define SubModel**

Submodeling option > **On**

Push boundary node flag > **No**

Geo Panel: Analysis > STATIC > **Run Static Analysis**

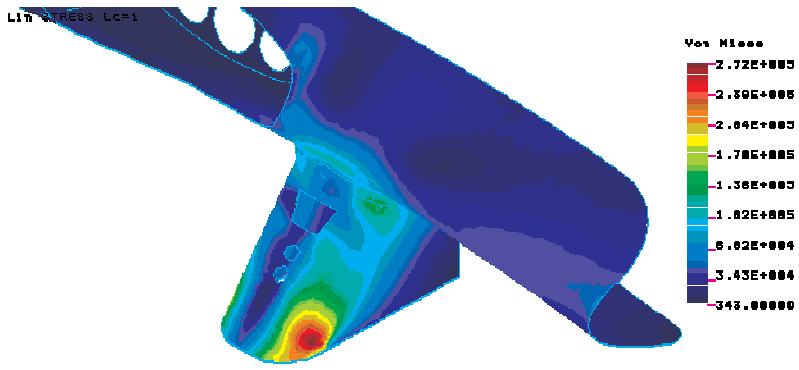
(Run analysis on the submodel)

Geo Panel: Control > SELECT > **Initialize**
Entity name > **Elements**
... ...
Accept Defaults

Geo Panel: Results > PLOT > **Stress**
Load case number > **1**
Component > **VON**
Layer number > **1**
Coordinate system > **0**
Stress Flag > **Nodal Stress**
Face of element > **Top Face**

Select Contour and Accept Defaults

Figure 11-34. Local von Mises Stresses for the Submodel (refined area is the actual submodel)



The maximum von Mises stress increased from the initial value of 2.61E5 psi to 2.72E5 psi.

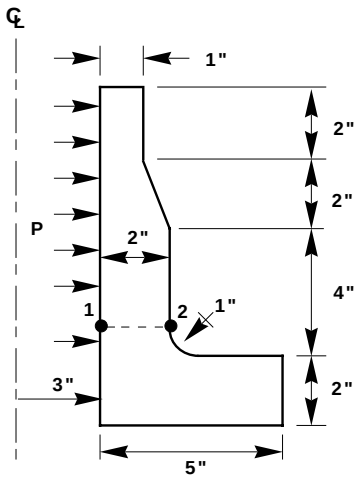
Example on ASME Stress Requirement

The stresses are reevaluated in the following example on the bases of the redundant forces and moments along a section associated with shell-type structures in order to be able to compare them with allowable stresses, Sm, 1.5 Sm and 3 Sm, specified by the ASME code section III and in accordance with Gordon, 1976 (refer to Chapter 2 *Mathematical Backgrounds*, as well as to Appendix C).

Example ASME.GEO - ASME Code Stress Evaluation

The input for this problem is provided to you in file ASME.GEO. The stresses are evaluated at the two ends of a section defined on a cross-sectional slice of an axisymmetric nozzle subjected to internal pressures according to the ASME code stress evaluations. This examples is intended to show the general use of this capability. However, in a typical problem you may need to consider several sections in order to find the maximum stresses. A section is defined by the two nodes at its ends. These nodes could be defined any where within the model and are not confined to the FEA mesh (i.e., they do not have to be associated with any elements). To find a proper orientation for sections at irregular areas such as in the nozzle to shell junctions please refer to Appendix C.

Figure 11-35. A Section Defined Through



In the following only the commands related to ASME code stress requirement are discussed.

In order to use this option, activate the ASME flag in the **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) command:

Geo Panel: Analysis > STATIC > **Stress Analysis Options**
AISC code check > **No**
... ...
ASME code check flag > **Yes**

Geo Panel: Analysis > STATIC > ASME CODE > **Define ASME Section**
ASME section > **1**
First node > **55**
Second node > **75**
Number of points > **25**
Radius of curvature: 0=plane > 0 axisym -1 straight > **-1**

Thickness direction bending stress flag > **Yes**

In the above command, the section is labeled one. You may define up to 300 sections at different locations of the structure. By the entries for second and third prompts, the section is defined at two ends by nodes 55 and 75. By choosing the default as 25 points for the fourth prompt, the program is instructed to consider only 25 integration points along the section. However, for a section with a very large stress gradient you may consider up to 65 points (i.e., you may check stress variation by plotting von Mises stresses along the section using the **LSECPLOT** (Results > PLOT > **Path Graph**) command).

The fifth prompt, "Radius of curvature" indicates whether the computation is considered for the plane type of problems (such as plane stress, plane strain or for axisymmetric problems approximated by Cartesian formulation) or true axisymmetric formulation such as the one considered in the present example. If you consider the structure at section 1 to be locally a cylinder with straight walls (inner and outer walls), then it is proper to consider the radius of curative to be infinite (see Appendix C, Subsection "*Axisymmetric Formulation*" for definition of the curvature-radius). This can be represented as -1 or equivalently a large number for that prompt. However, if you want to consider the curvature at the lower part of the section, you may do so, by choosing the radius of curvature of the average mid-wall centerline which in this model can be approximated to be equal to 2 inches.

The last prompt is used to consider or ignore the thickness direction bending stresses (see Appendix C for more details). You may use "**A_LIST, STRESS**" (Analysis > **List Analysis Option**) to check the status of this option as well as "**ASMESECLIS** and **ASMESECDEL**" (Analysis > STATIC > ASME CODE > **List ASME Section**, **Delete ASME Section**) to list or delete the sections. After running the analysis, the results are printed at the end of output file which you may compare with the allowable stresses required by the ASME code.

Since all the computations for the ASME code stresses are performed during the stress calculation, you do not need to perform the displacement calculation part of the solution, every time you need to rerun for modified or added sections. In that case simply reissue the **R_STRESS** (Analysis > STATIC > **Run Stress Analysis**) command. Furthermore, if you wish to define the section along a path which do not coincide with the natural boundary of the element mesh, you may do so by first defining two nodes anywhere within the model and then associate them to the two ends of the desired section. In addition to the static analysis, the ASME code stress evaluation option is also available with the Advanced Dynamic module for the time history analysis.




Example on the Use of Multiple Thermal Load Cases

Thermal effects can be considered in all of the primary load cases and may be input directly by the user as nodal temperatures or may be read from a prior heat transfer analysis. In the following example the application of both types of thermal loadings is illustrated.

Example TEMP1.GEO - Multiple Thermal Load Cases

The input for this problem is provided to you in file TEMP1.GEO. The model of an axisymmetric nozzle is subjected to five different loading environments, each represented by a load case as follows:

a. Load case 5:

   Concentrated force at the tip of the nozzle plus nodal temperatures from step 6 of a prior transient heat transfer analysis.

b. Load case 10:

   Nodal temperatures defined at certain nodes.

c. Load case 20:

   Nodal temperatures defined at certain nodes.

d. Load case 30:

   Nodal temperatures from step 20 of the prior transient heat transfer analysis.

e. Load case 40:

   Nodal temperatures from step 10 of the prior transient heat transfer analysis.

Figure 11-36. Axisymmetric Nozzle

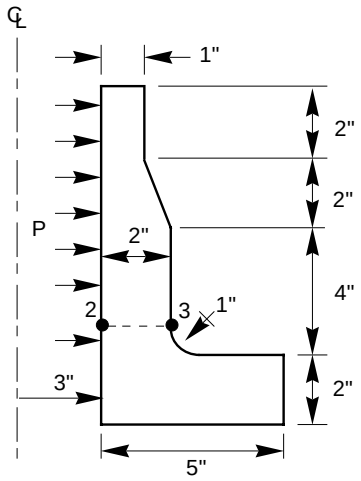
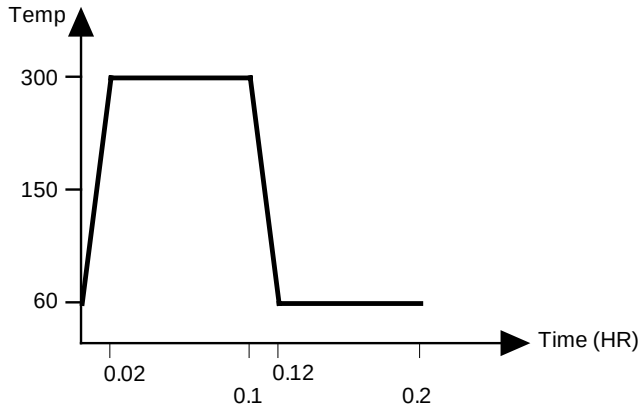


Figure 11-37. Nozzle Heated by Fluid



Assuming the nozzle is heated by a fluid flow with temperature variation as shown in the figure above, the first step is to solve the transient heat transfer problem using HSTAR or FFE Thermal. To review the commands pertinent to thermal part of the problem, refer to the TEMP1.GEO file. After you solve the thermal problem, activate the thermal loading flag:

Geo Panel: Analysis > STATIC > **Static Analysis Options**
Loading flag > **T**
... ...
Accept Defaults

To define load case number 5:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **LC**

Load case set number > **5**

(Activate load case 5)

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **101**
Force label > **FX: X force**
Value > **100**
Ending node > **101**
Increment > **1**

(Apply a nodal force of 100 at node 101)

Geo Panel: LoadsBC > LOAD OPTIONS > **Read Temp as Load**
Time step label > **6**
Load case number > **5**

(Assign temperature profile from time step 6 of heat transfer analysis to load case 5)

To define load case number 10:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **LC**

Load case set number > **10**

(Activate load case 10)

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > **Define Nodes**
Beginning node > **101**
Value > **200**
Ending node > **104**
Increment > **1**

(Define a temperature of 200 at nodes 101 through 104)

To define load case number 20:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **LC**

Load case set number > **20**

(Activate load case 20)

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > **Define Nodes**
Beginning node > **101**
Value > **400**
Ending node > **104**
Increment > **1**

(Define a temperature of 400 at nodes 101 through 104)

To define load case number 30:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **LC**

Load case set number > **30**

(Activate load case 30)

Geo Panel: LoadsBC > LOAD OPTIONS > **Read Temp as Load**
Time step label > **20**
Load case number > **30**

(Add the thermal effect from the heat transfer time step 20)

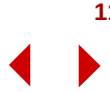
To define load case number 40:

Geo Panel: LoadsBC > LOAD OPTIONS > **Read Temp as Load**
Time step label > **10**
Load case number > **40**

(To consider the thermal effect from the heat transfer time step 10)

Once all the load cases are defined, you may run the static analysis by issuing the **R_STATIC** (Analysis > STATIC > **Run Static Analysis**) command. Postprocessing will be available for all load cases as usual.






12 Multidisciplinary Analysis

Introduction

In COSMOS/M, the pre- and postprocessing module, GEOSTAR, provides a *one-screen* solution to all analysis modules. You can build a finite element model and perform different types of analysis on the same model within GEOSTAR. In addition, it is possible to consider the *interaction* of different analysis modules such that the analysis results of one module can be transferred as load to another. This type of interaction is very useful for many practical analysis problems. For example, you can perform a transient heat transfer analysis using HSTAR, and transfer the output as thermal loads to the Basic System using the command **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**). This chapter considers the interaction of advanced modules with the Basic System only, and for a complete picture of interactions between various analysis modules of COSMOS/M, refer to Chapter 3, *Exploring GEOSTAR*, of the COSMOS/M User Guide.

The term *multidisciplinary analysis* in this manual refers to the interaction of electromagnetic, heat transfer, nonlinear, and fluid flow analyses with linear static, buckling, and modal analysis available in the Basic FEA System. The interaction of advanced modules, ESTAR, HSTAR, NSTAR, and FLOWPLUS with the STAR and DSTAR modules of the Basic System is *unidirectional*. You need to first perform the required finite element analysis using an advanced analysis module. After successful completion of the analysis, the results are transferred using one of the *read* commands as equivalent mechanical or thermal loads to the Basic System for further analysis.

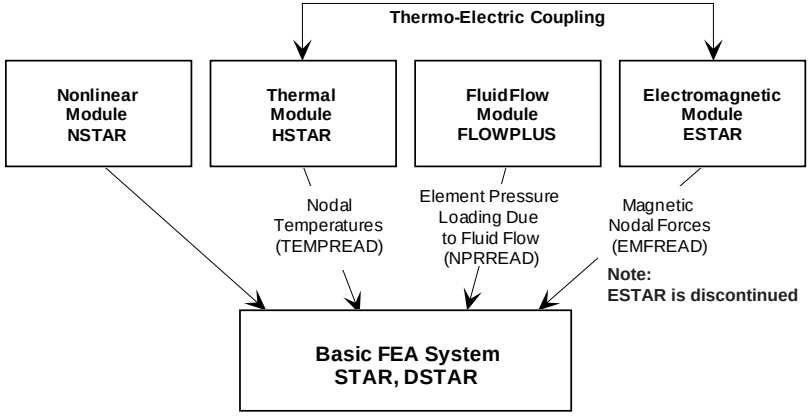
This chapter presents a detailed discussion on how to perform a multidisciplinary analysis in the Basic System. The required commands and procedures are explained through examples which include all relevant steps with figures and illustrations.

It is also possible to combine the results of advanced dynamics analysis (using the ASTAR module) with the results of the Basic System analysis during the dynamic analysis or afterward, during the postprocessing.

Multidisciplinary Analysis in COSMOS/M

The interaction of advanced analysis modules with the Basic System is represented in the figure below. The terms thermoelastic analysis, quasi-static analysis, and magnetoelastic analysis, refer to the types of analyses you can perform as a result of interaction with the Basic System. In order to perform these analyses, you *must* have the respective advanced modules in addition to the Basic System. Note that there is a *unidirectional* coupling between thermal and electromagnetic modules and it is specified using **A_MAGNETIC** (Analysis > ELECTRO MAGNETIC > **Analysis Options**) or **A_THERMAL** (Analysis > HEAT TRANSFER > **Thermal Analysis Options**) commands. The temperature results can be then moved to the Basic System for thermoelastic analysis.

Figure 12-1. Interaction of Advanced Analysis Modules with the Basic System



This feature can be used to analyze problems such as thermal strains due to Joule heating. For frequency and buckling analysis, you may generate the stiffness matrices based on their material nonlinearities by using the NSTAR module. Then, using the DSTAR module, you can evaluate the frequency or buckling characteristics of your model. Use the appropriate flags in **A_FREQUENCY** (Analysis > FREQUENCY/ BUCKLING > **Frequency Options**) or **A_BUCKLING** (Analysis > FREQUENCY/ BUCKLING > **Buckling Options**) commands for this purpose. See also the NSTAR portion of the Advanced Modules manual for more information.

The command **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) assigns nodal temperatures from a steady state or transient heat transfer analysis at a specified time step to load cases of a linear static analysis, whereas the command **NPRREAD** (LoadsBC > LOAD OPTIONS > **Read Fluid Pressure**) applies the pressure calculated by the fluid flow analysis at the given time step on the faces of elements associated with the specified curves. The **EMFREAD** (LoadsBC > LOAD OPTIONS > **Read Emag Force**) command is used for magneto-structural coupling, and it provides the Basic System with the magnetic nodal forces. The table below summarizes the functions of these commands and the corresponding sections where they are discussed in more detail with examples. All commands in this table are found in LoadsBC menu, and LOAD OPTIONS submenu.

Table 12-1. Commands for Performing Multidisciplinary Analysis

Type of Data Transfer to Basic System	Command (Cryptic)	See Section...
Heat Transfer (HSTAR) (STAR, DSTAR)	TEMPREAD	Thermoelastic Analysis Using HSTAR and STAR. Example TEMPR1 - Thermal Stress Analysis of a Heat Exchanger.
Fluid Flow (FLOWPLUS) (STAR, DSTAR)	NPRREAD	Quasi-static Stress Analysis using FLOWPLUS and STAR.
Electromagnetics (ESTAR) (STAR, DSTAR)	EM_OUTPUT EMFREAD	Magnetoelastic Analysis using ESTAR and STAR Example EM16 - Magnetoelastic Analysis of Long Cylinder.

All analysis modules (*except* the thermal module, HSTAR) write the results of analysis into an output file with extension OUT. The results of thermal analysis are written to a file with extension TEM. However, the contents of the output files are *replaced* with the results of last performed analysis by default. Therefore, if you like to preserve the results output from the previous execution, you need to use the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command.

✍ The input files of examples discussed in this chapter are provided to you in a compressed form in the file, PROBLEMS.LZH, installed in the COSMOS/M directory. However, it is instructive to practice these examples step by step as presented in this chapter.

As mentioned earlier, it is possible to combine the results of an advanced dynamics analysis (using the ASTAR module) with the results of the Basic System analysis, ***only*** during postprocessing. This task can be accomplished using the command **READ_PDRESP** (**Read Post-Dyn Response**) from the Results menu. This command reads displacements and stresses for a specified time step (or frequency) of an advanced dynamic analysis and stores it as a secondary load case for postprocessing in the Basic System. The secondary load case can be combined with primary load cases of the Basic System analysis using the command **LCCOMB** (Results > **Combine Load Case**). The application of the **READ_PDRESP** (Results > **Read Post-Dyn Response**) command requires the advanced dynamics analysis results to be in the same database as that of the Basic System analysis.

The following sections discuss the application of the commands for unidirectional interaction between advanced analysis modules and the Basic System with detailed examples.

Thermoelastic Analysis Using HSTAR and STAR

The thermal module HSTAR can be used to perform linear and nonlinear heat transfer analysis of conduction, convection and radiation problems. Irrespective of the type of thermal analysis you perform, it is possible to transfer the results as thermal loads to the Basic System. The **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) command associates nodal temperatures from a transient heat transfer analysis at a specified time step and to a load case of the Basic System. You can specify the time step number of a transient thermal analysis at which nodal temperatures are to be used. For steady state heat transfer problems, you should consider 1 for the time_step. You may consider up to 50 thermal

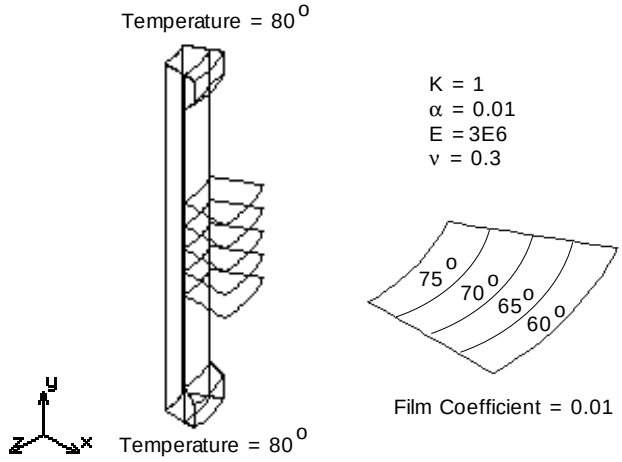
load cases in your analysis. Please refer to Chapter 7, *Loads and Boundary Conditions*, for more details.

✍ You need to activate the special thermal loading flag using the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command in order to consider thermal loading.

Example TEMPR1 - Thermal Stress Analysis of a Heat Exchanger

This example (file name, TEMPR1.GEO) deals with thermal stress analysis of a heat exchanger consisting of a 30° pipe section with fins on the outer surface for cooling. In the figure below, the material properties as well as the thermal boundary conditions for heat transfer analysis are shown. The figure also shows an enlarged view of a fin with varying ambient temperatures. For structural analysis, it is assumed that the top and bottom surfaces of the pipe section are fixed, and the only loading imposed is thermal, read from heat transfer analysis.

Figure 12-2. Geometry and Thermal Boundary Conditions and Loads of the Cooling System



The geometry of the heat exchanger can be easily created by first constructing the cross section and sweeping it by an angle of 30° about the y-axis. The procedure used here for building the model is not necessarily the optimum one, as the emphasis is to show how the thermal module and the Basic System interact.

To start with, after entering GEOSTAR, establish a working plane as instructed below:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line Style >**Solid**

Next, establish keypoints for surfaces and lines using the **PT** command with coordinates as shown below:

Geo Panel: Geometry > POINTS > **Define**
Keypoint > **1**
XYZ-coordinate of Keypoint 1 > **25,150,0**

Repeat the above command for points 2 through 20 at the following coordinates:

Point No.	XYZ Coordinates
2	40,150,0
3	40,140,0
4	35,135,0
5	25,0,0
6	40,0,0
7	40,10,0
8	35,15,0
9	35,75,0
10	55,75,0
11	35,65,0
12	55,65,0
13	35,85,0
14	55,85,0
15	35,95,0
16	55,95,0
17	35,55,0
18	55,55,0
19	35,150,0
20	35,0,0

Use the **SCALE** (Auto Scale icon) command if necessary to see all keypoints you just created. Next, using these keypoints, you need to create three surfaces and five lines. The surfaces, when swept about the y-axis, form volumes representing the pipe section whereas the lines form the cooling fin surfaces after sweeping. The surfaces and lines of the cross section are created as illustrated below:

Geo Panel: Geometry > SURFACES > **Define by 4 Pt**
Surface > **1**
Keypoint 1 > **5**
Keypoint 2 > **20**
Keypoint 3 > **19**
Keypoint 4 > **1**
Underlying Surface >

Repeat the above command to create surface 2 using keypoints 19,2,3,4 and surface 3 using keypoints 8,7,6,20.

Geo Panel: Geometry > CURVES > **Line with 2 Pts**
Curve > **13**
Keypoint 1 > **15**
Keypoint 2 > **16**

Repeat the above command to create curve 14 between keypoints 13,14, curve 15 between 9,10, curve 16 between 11,12, and curve 17 between 17,18.

Clear the screen and change the view to three dimensional one as illustrated below. Use the **VLSWEEP** (Geometry > VOLUMES > GENERATION MENU > **Sweeping**) command to sweep surfaces 1 through 3 about the y-axis, resulting in the formation of three volumes. You can next sweep curves 13 through 17 about the y-axis using the command **SFSWEEP** (Geometry > SURFACES > GENERATION MENU > **Sweeping**) to form the fin surfaces, as illustrated below:

Geo Panel: Geometry > VOLUMES > GENERATION MENU > **Sweeping**
Beginning surface > **1**
Ending surface > **3**
Increment > **1**
Axis symbol > **Y**
Angle of the arc > **30**
Number of segments > **1**

Geo Panel: Geometry > SURFACES > GENERATION MENU > **Sweeping**
Beginning curve > **13**
Ending curve > **17**
Increment > **1**
Rotation/sweep Axis > **Y**
Angle of the arc > **30**
Number of segments > **1**

The volumes representing the pipe section will be meshed with solid elements where as the cooling fins are modeled with quadrilateral shell elements; see illustration below:

Geo Panel: Propsets > **Element Group**
Element group > **1**
Element category > **Volume**
Element Type (for volume) > **SOLID: 8- to 20-node 3D solid element**

... Accept Defaults ...

Geo Panel: Meshing > PARAMETRIC MESH > **Volumes**
Beginning volume > **1**
Ending volume > **1**
Increment > **1**
Number of nodes per element > **8**

Number of elements on 1st curve > **3**

Number of elements on 2nd curve > **30**

Number of elements on 3rd curve > **3**

... Accept Defaults ...

Geo Panel: Meshing > PARAMETRIC MESH > **Volumes**
Beginning volume > **2**
Ending volume > **3**
Increment > **1**
Number of nodes per element > **8**

Number of elements on 1st curve > **2**

Number of elements on 2nd curve > **3**

Number of elements on 3rd curve > **3**

... Accept Defaults ...

The elements in the fins will be associated with element group and section constant set number 2:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **RC**

Real constant set number > **2**

Geo Panel: Propsets > **Element Group**
Element group > **2**
Element category > **Area**
Element Type (for area) > **SHELL4: 4-node thin shell element**

... Accept Defaults ...

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **19**
Ending surface > **23**
Increment > **1**
Number of nodes per element > **4**

Number of elements on 1st curve > **4**

Number of elements on 2nd curve > **3**

... Accept Defaults ...

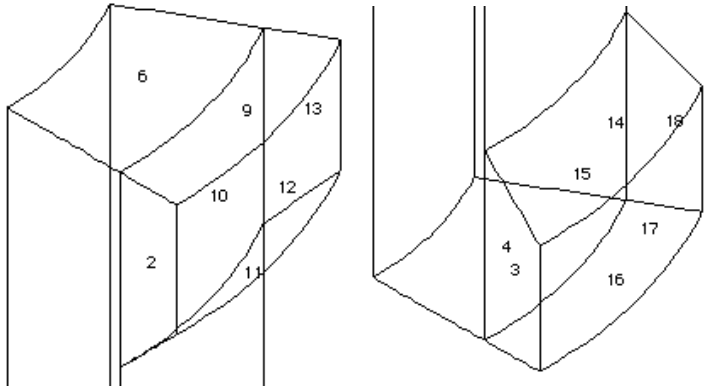
The nodes at the common boundaries need to be merged in order to satisfy compatibility, using the command **NMERGE** (Meshing > NODES > **Merge**). The **NCOMPRESS** (Edit > COMPRESS > **Nodes**) command removes the numbering gaps in nodes, resulting from merging operations.

Geo Panel: Meshing > NODES > **Merge**
Total number of nodes merged = 52

Geo Panel: Edit > COMPRESS > **Nodes**
Number of Nodes compressed = 52

You can next proceed to defining the thermal boundary conditions. Clear the screen and issue **ACTNUM,SF,1** (Control > ACTIVATE > **Entity Label**) and **SFPLOT** (Edit > PLOT > **Surfaces**) commands to plot surfaces with numbers. Zoom-in using the icon on the top and bottom surfaces of the pipe section (separately) on which you need to impose a temperature of 80° . As shown in the figure below, these surfaces are numbered 6 and 9 at the top, and 4 and 16 at the bottom.

Figure 12-3. Enlarged Views of the Top and Bottom Surfaces of the Pipe



Select the **NTSF** (LoadsBC > THERMAL > TEMPERATURE > **Define Surfaces**) command to apply temperatures at all nodes associated with the top and bottom surfaces, as illustrated below:

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > **Define Surfaces**
Beginning surface > **6**
Value > **80**
Ending surface > **9**
Increment > **3**

Beginning surface > **4**
Value > **80**
Ending surface > **16**
Increment > **12**

Geo Panel: LoadsBC > THERMAL > CONVECTION > **Define Elements**
Beginning Element > **310**
Convection coefficient > **0.01**
Ambient temperature > **60**
Face number > **5**
Ending Element > **366**
Increment > **4**
Time curve for ambient temperature > **0**

Beginning Element > **309**
Convection coefficient > **0.01**
Ambient temperature > **65**
Face number > **5**
Ending Element > **365**
Increment > **4**
Time curve for ambient temperature > **0**

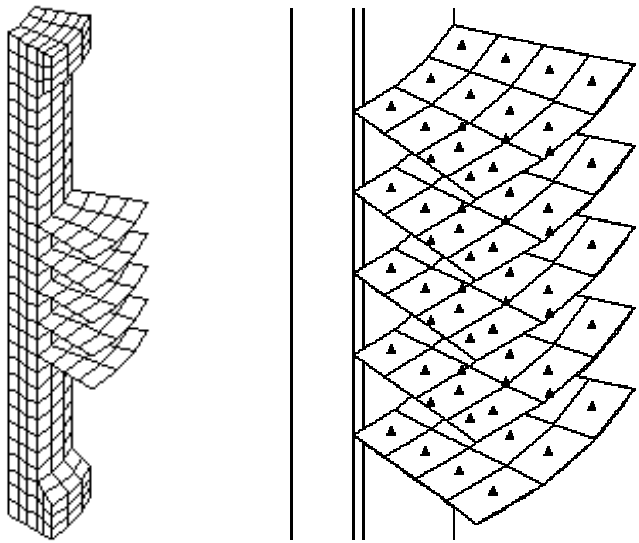
Beginning Element > **308**
Convection coefficient > **0.01**
Ambient temperature > **70**
Face number > **5**
Ending Element > **364**
Increment > **4**
Time curve for ambient temperature > **0**

Beginning Element > **307**
Convection coefficient > **0.01**
Ambient temperature > **75**

Face number > **5**
Ending Element > **363**
Increment > **4**
Time curve for ambient temperature > **0**

The elements on the cooling fins are numbered from 307 to 366. Each fin has four rows of elements corresponding to the four ambient temperatures imposed. You can activate the element numbering using the command **ACTNUM,EL,1** (Control > ACTIVATE > **Entity Label**) and plot these elements using the command **EPLOT,307,366** (Edit > PLOT > **Elements**). Zoom-in on these elements to obtain a clear view of the element numbers. The outermost row of elements are numbered in such a way that you can apply film coefficients and ambient temperatures on all fins by incrementing the element numbers by 4. The illustration above shows the application of film coefficients and ambient temperatures for all fins using the **CEL** (LoadsBC > THERMAL > CONVECTION > **Define Elements**) command. The element face numbers (5 for the top face, as input above) are explained in more detail in the Chapter 4, *Element Library*, of the COSMOS/M User Guide. The figure below shows the finite element mesh of the cooling system as well as an enlarged view of the fins with convection coefficient definitions.

Figure 12-4. Finite Element Mesh and Enlarged View of the Cooling Fins



The definition of thermal conductivity and sectional data for the shell elements illustrated below, will complete the input for heat transfer analysis:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material property name > **KX; X Thermal conductivity**
Property value > **1**

Geo Panel: Propsets > **Real Constant**
Associated element group > **2**
Real constant set > **2**

Start loc. of the real const. > **1**
No. of real const. to be entered > **6**

RC1: Thickness > **0.1**
... Accept Defaults ...

Use the command **R_THERMAL** (Analysis > HEAT TRANSFER > **Run Thermal Analysis**) to perform a steady-state heat transfer analysis. After the analysis is successfully executed, the control returns to the GEOSTAR screen. To consider thermal loading, you need to associate the results from the thermal steady-state analysis with the desired load case using the **TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) command. The commands **ACTTEMP** and **TEMPPLOT** (Results > PLOT > **Thermal**) can be successively issued to graphically view the temperature results that will be considered for the Basic System. The thermal loading considered is equivalent to applying nodal temperatures from the thermal analysis to all nodes using the **NTND** (LoadsBC > THERMAL > TEMPERATURE > **Define Nodes**) command. If you activate the input print flag using the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command, the thermal loading considered will be written to the output file.

Geo Panel: LoadsBC > LOAD OPTIONS > **Read Temp as Load**
Time step label > **1**
Load case number > **1**

For thermal stress analysis, the top and bottom surfaces are fixed in all directions using the **DSF** (LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Surfaces**) command. Additional material properties required for stress analysis are the elastic modulus and Poisson's ratio, defined as shown below:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Surfaces**
Beginning surface > **4**
Displacement label > **ALL; All 6 DOF**
Value > **0.0**
Ending surface > **16**
Increment > **12**

Beginning surface > **6**
Displacement label > **ALL; All 6 DOF**
Value > **0.0**
Ending surface > **9**
Increment > **3**

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material property name > **EX: Elasticity modulus 1st dir**
Property value >

Material property name > **ALPX: Therm expansion coeff 1st dir**
Property value > **0.01**

In COSMOS/M, thermal and gravitational loading are treated as special loading (see Chapter 7, *Loads and Boundary Conditions*), as such they need to be activated using the **A_STATIC** (Analysis > STATIC > **Static Analysis Options**) command before performing an analysis. In addition, using the same command, the soft spring option was activated as the element types used in this example have incompatible displacement fields. Alternately, you can restrain the θz component at any node of a shell element in each fin to provide stability. The addition of soft springs (explained in Chapter 2, *Mathematical Background*) does not significantly alter the results.

Geo Panel: Analysis > STATIC > **Static Analysis Options**
Loading flag > **T: Thermal loading**
In-plane effect flag > **No**
Soft Spring flag > **Yes**
... Accept Defaults ...

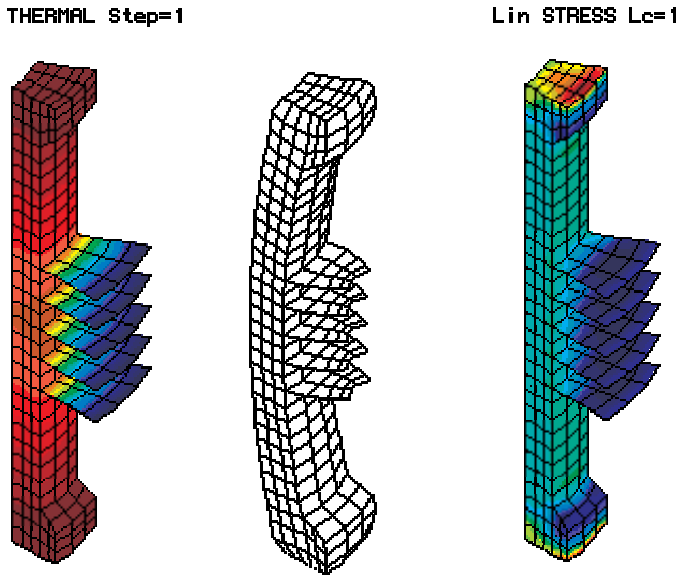



You can now proceed with thermal stress analysis using the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**). After the execution is successfully completed, you can proceed with postprocessing. Since static stress analysis was last performed, it becomes the active analysis type for postprocessing. You can postprocess the results of heat transfer analysis by first activating it as illustrated below:

Geo Panel: Results > SET UP > **Set Postprocess Type**
Analysis type > **Thermal**

You can plot the temperature contours using the command **TEMPPLOT** (Results > PLOT > **Thermal**), or list heat transfer analysis results using **TEMPLIST** (Results > LIST > **Thermal Results**). To view stress analysis results, repeat the **ACTPOST** (Results > SET UP > **Set Postprocess Type**) command and specify 0 for linear static analysis. The commands **DEFPLOT** (Results > PLOT > **Deformed Shape**) and **ACTSTR** with **STRPLOT** (Results > PLOT > **Stress**) can be used to plot the deformed shape and stress contours respectively. The figure below shows the deformed shape as well as temperature and stress contours.

Figure 12-5. Temperature Contour, Deformed Shape, and Stress Contour Plots



Magnetoelastic Analysis Using ESTAR and STAR

Magneto-solid mechanics is the study of the effect of magnetic forces on the deformations and stresses of solids. In *magnetoelastic* analysis, the treatment of these effects is linear. The need for magnetoelastic analysis arises from a consistent increase in applications ranging from switches, generators, superconducting motors, fusion reactors, magnetic levitation trains, MHD (magnetohydrodynamics) devices, and many others. Using the Basic System, the types of analysis you can perform include magnetoelastic stress, and magnetoelastic stability problems.

The observable quantities in magnetoelastic analysis include the displacements (**u**), strains ($\boldsymbol{\varepsilon}$), stresses ($\boldsymbol{\sigma}$), electric current (**J**), magnetic flux (**B**), and temperature (**T**). The effect of electromagnetic fields on the mechanical behavior include (Moon, 1984):

- Change in static stresses due to electromagnetic body forces.
- Generation of angular moments due to body couples.
- Generation of thermal energy due to Joule heating or ferromagnetic hysteresis.
- Change in strains due to magnetic fields (magnetostriction).
- Joule-heating-induced thermoelastic strains.

Of the above mentioned effects, only the first one is accessible directly by using ESTAR and the Basic System. Note that the generation of thermal energy due to Joule heating can be considered by performing a coupled thermo-electric analysis in COSMOS/M. The last effect dealing with thermoelastic strains induced due to Joule heating can be considered by first performing a coupled thermo-electric analysis, and subsequently transferring the results as thermal loading (**TEMPREAD** (LoadsBC > LOAD OPTIONS > **Read Temp as Load**) command) for structural analysis in the Basic System.

Computation of Electromagnetic Forces

In COSMOS/M, there are two approaches to computing electromagnetic forces:

- Lorentz law
- Virtual work method

The first method which computes Lorentz forces is only applicable for current carrying conductors and it is based on the following equation:

$$F = \int J \times B \, dv \qquad (12\text{-}1)$$

where X represents the vector cross product. The virtual work technique can be used for obtaining the forces on ferromagnetic solids under externally applied field as well as current carrying conductors. The details of this technique are rather involved and they can be found in the literature (see Advanced Modules Manual). In order to calculate the forces on an object, the user needs to identify the region containing the object and assign virtual displacement of a unit value to the elements within this object. The rest of the region in the model is assigned a virtual displacement of zero (default for magnetic elements). It should be noted that the object *must* be surrounded by a layer of air in order to use this method.

The **EM_OUTPUT** (Analysis > ELECTRO MAGNETIC > **Output Options**) command is used to specify one of the above methods of force calculation *before* executing an analysis using ESTAR. The types of force calculation available are Lorentz forces (default), and the computation of forces using the virtual work method.

✍ The Lorentz force calculation is *only* available for 2D and axisymmetric models, and the virtual work option is *not* available for models with 3D current sources.

After the analysis is successfully completed using ESTAR, the electromagnetic forces are transferred as nodal point loads to the Basic FEA System (and other modules) using the **EMFREAD** (LoadsBC > LOAD OPTIONS > **Read Emag Force**) command. You need to specify the time step number at which the magnetic forces are to be transferred to linear static analysis. The forces transferred can be plotted and listed using the commands **FPLOT** and **FLIST** (LoadsBC > STRUCTURAL > FORCE > **Plot**, **List**).

Modeling Hints

The following are some hints you can use in modeling magneto-structural problems:

- To start with, the finite element model *must* consist of only those elements which are valid for electromagnetic analysis using ESTAR. The valid element types currently supported in the electromagnetic module and their equivalent elements in the Basic System are shown below:

ESTAR	Basic System
MAG2D	PLANE2D
MAG3D	SOLID
TETRA4	TETRA4, TETRA4R
TETRA10	TETRA10

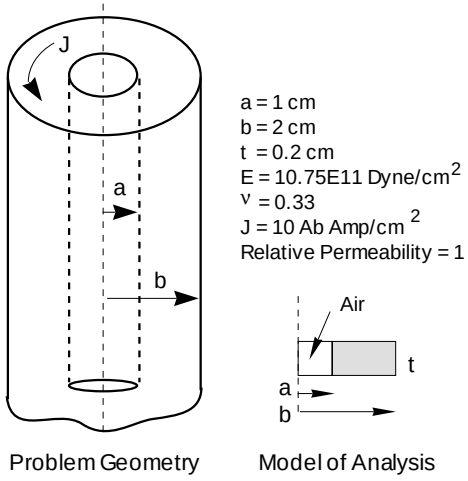
- After successful completion of the electromagnetic analysis, *delete* parts of your mesh that are *not* required for structural analysis. These parts may include air gaps, fluids surrounding magnets, and others.
- If the dead loads due to the deleted parts of the mesh are significant and are required to be considered in structural analysis, you can approximate and define them separately as loads.
- Redefine the element type for the remaining parts of your mesh using the equivalent structural element indicated in the above table and make sure the element type selected conforms to the existing nodes.
- Use the **EMFREAD** (LoadsBC > LOAD OPTIONS > **Read Emag Force**) command to transfer electromagnetic forces on the structure. The commands **FPLOT** (LoadsBC > STRUCTURAL > FORCE > **Plot**) or **FLIST** (LoadsBC > STRUCTURAL > FORCE > **List**) can be used to plot or list the forces so transferred. Note that due to numerical tolerances adopted, you may some times see components of forces that are not significant when **FPLOT** (LoadsBC > STRUCTURAL > FORCE > **Plot**) command is issued for all components.

Example EM16 - Magnetoelastic Analysis of a Long Cylinder

This example (file EM16.GEO) deals with stress distribution in a long thick solenoid due to magnetic forces resulting from a uniform distribution of circumferential current. The solenoid is assumed to be made of isotropic elastic material. Since the solenoid is very long, the length in the axial (vertical) direction is assumed to be infinite. For convenience, a 0.2 cm length of the cylinder is considered for analysis as shown in the figure below. To model the infinite length effects, the axial displacements are set to zero at the top and bottom faces of the model for analysis. It is also assumed that the magnetic field outside the solenoid is zero, and the only non-zero component of the magnetic flux is in the axial

direction. Dirichlet boundary condition of zero magnetic potentials are imposed at the center r = 0, and, at r = 2, the magnetic potentials set to be equal. All other boundaries are subjected to Neumann boundary conditions automatically.

Figure 12-6. Geometry of the Solenoid for Magnetoelastic Analysis



After entering GEOSTAR, establish a working plane as instructed below:

Geo Panel: Geometry > GRID > **Plane**
Rotation/sweep Axis > **Z**
Offset on Axis > **0.0**
Grid Line Style > **Solid**

Change the view to XY using the Viewing icon. Next, establish keypoints for creating surfaces using the **PT** (Geometry > POINTS > **Define**) command with coordinates as shown below:

Geo Panel: Geometry > POINTS > **Define**
Keypoint > **1**
XYZ-coordinate of Keypoint 1 > **0,0,0**

Repeat the above command to create the following keypoints:

Point No.	XYZ Coordinates
2	1,0,0
3	1,0.2,0
4	0,0.2,0
5	2,0,0
6	2,0.2,0

Use the **SCALE** command (or Auto Scale icon) if necessary to clearly see all keypoints you just created. Next, using these keypoints, you need to create two surfaces:

Geo Panel: Geometry > SURFACES > **Define by 4 Pt**
Surface > **1**
Keypoint 1 > **1**
Keypoint 2 > **2**
Keypoint 3 > **3**
Keypoint 4 > **4**
Underlying surface > **0**

Geo Panel: Geometry > SURFACES > **Define by 4 Pt**
Surface > **2**
Keypoint 1 > **2**
Keypoint 2 > **5**
Keypoint 3 > **6**
Keypoint 4 > **3**
Underlying surface > **0**

The first surface represents the cross section of the air gap, and the second surface represents that of the solenoid. These two surfaces will be modeled using one row of elements, with fewer elements in the air gap.

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **1**
Ending surface > **1**
Increment > **1**
Number of nodes per element > **4**

Number of elements on 1st curve > **8**
Number of elements on 2nd curve > **1**

... Accept Defaults ...

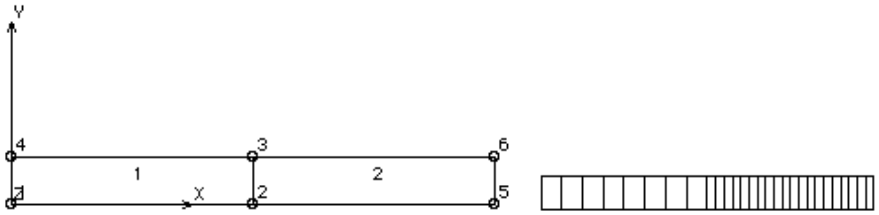
Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
Beginning surface > **2**
Ending surface > **2**
Increment > **1**
Number of nodes per element > **4**

Number of elements on 1st curve > **20**
Number of elements on 2nd curve > **1**

... Accept Defaults ...

The surface plot and finite element mesh are shown in figure below:

Figure 12-7. Surface and Finite Element Mesh Plots of the Solenoid Model



Activate the label display of curves using the command **ACTNUM,CR,1** (Control > ACTIVATE > **Entity Label**) (this can also be done in the status table using **STATUS1** icon). Clear the screen and plot all curves, using Edit > PLOT > **Curves**.

At r = 0 which corresponds to curve number 3, the magnetic potential is set to zero using the command **NPCR** (LoadsBC > E-MAGNETIC > MAGNETIC POTENTIAL > **Define by Curves**) as shown below:

Geo Panel: LoadsBC > E-MAGNETIC > MAGNETIC POTENTIAL > **Define by Curves**
Beginning Curve > **3**
Value > **0**
Ending Curve > **3**
Increment > **1**

Clear the screen and activate node numbers using the **ACTNUM** (Control > ACTIVATE > **Entity Label**) command. Plot all nodes using **NPLOT** (Edit >

PLOT > **Nodes**) and zoom-in on the right edge of the finite element model. At this edge (r = 2), the magnetic potentials are set to be equal using the command **MCPDEF** (LoadsBC > E-MAGNETIC > **CurntSrc/MagCoupling** > **Define Magnetic Coupling**) as shown below:

Geo Panel: LoadsBC > E-MAGNETIC > **CurntSrc/MagCoupling > Define Magnetic Coupling**
Pattern 1 Beginning Node > **39**
Pattern 1 Ending Node > **39**
Pattern 1 Increment > **1**
Pattern 2 Beginning Node > **60**
Pattern 2 Ending Node > **60**
Pattern 2 Increment > **1**

Clear the screen and issue **SFPLOT** (Edit > PLOT > **Surfaces**) command. The solenoid is subjected to a uniform distribution of current which can be applied using the command **JESF** (LoadsBC > E-MAGNETIC > ELEMENT CURRENT > **Define by Surfaces**) on surface 2:

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT CURRENT > **Define by Surfaces**
Beginning Surface > **2**
Value > **10**
Ending Surface > **2**
Increment > **1**

You can now proceed to defining the material properties and element group definitions as illustrated below:

Geo Panel: Propsets > **Material Property**
Material property set > **1**
Material Property Name > **MPERM: Magnetic Permeability**
Property value > **1**

Geo Panel: Propsets > **Element Group**
Element Group > **1**
Element Name > **MAG2D: 2D magnetic element**

OP1: 2D Type > **Axisymmetric**
... Accept Defaults ...




The magnetic force calculation is specified as shown below:

Geo Panel: Analysis > ELECTRO MAGNETIC > **Output Options**
Magnetic field info flag 0=No N=steps > **1**
Postprocessing info flag > **Yes**
Force Calc. > **Standard**
... Accept Defaults ...

Since the units have been specified in CGS units, you need to use the **A_MAGNETIC** (Analysis > ELECTRO MAGNETIC > **Analysis Options**) command and use this system of units in analysis. With the finite model completed for electromagnetic analysis, you can proceed with analysis after merging nodes on the common boundary as illustrated below:

Geo Panel: Meshing > NODES > **Merge**

Geo Panel: Analysis > ELECTRO MAGNETIC > **Analysis Options**
Analysis option > **S; Magnetostatic**
Units > **CGS**
... Accept Defaults ...

Geo Panel: Analysis > ELECTRO MAGNETIC > **Run EMag Analysis**

After the electromagnetic analysis is successfully completed, you can postprocess various components of results (see Advanced Modules Manual for more information). Since the air gap is not required for stress analysis, you can delete the elements in surface 1 using the **MSFDEL** command, **MSFDEL,1,1,1;** (Edit > DELETE > **Element on Surface**). Ignore warning messages when you delete the elements in surface 1. The forces for stress analysis are transferred as follows:

Geo Panel: LoadsBC > LOAD OPTIONS > **Read Emag Force**
Time step label > **1**

You can use the commands **FPLOT** and **FLIST** (LoadsBC > STRUCTURAL > FORCE > **Plot**, **List**) to plot and list the forces transferred from the electromagnetic analysis. The top and bottom curves of surface 2 are constrained in the axial direction (y-axis) for stress analysis as illustrated below:

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define by Curves**
Beginning Curve > **5**
Displacement label > **UY: Y Translation**
Value > **0.0**
Ending Curve > **6**
Increment > **1**

The material properties for stress analysis are appended to material set 1 as follows:

Geo Panel: Propsets> **Material Property**
Material property set > **1**
Material Property Name > **EX: Elasticity modulus in X mat. dir.**
Property value > **10.76E11**

Material Property Name > **NUXY: XY Poisson's ratio**
Property value > **0.33**

For stress analysis, the equivalent element is PLANE2D with the axisymmetric modeling option activated as illustrated below:

Geo Panel: Propsets> **Element Group**
Element group > **1**
Element Name > **PLANE2D: 2D 4- 8 node plane element**

OP1:S/F flag > **Solid**
OP2:Integr Type > **QM6**
OP2:Type > **Axisymmetric**
... Accept Defaults ...

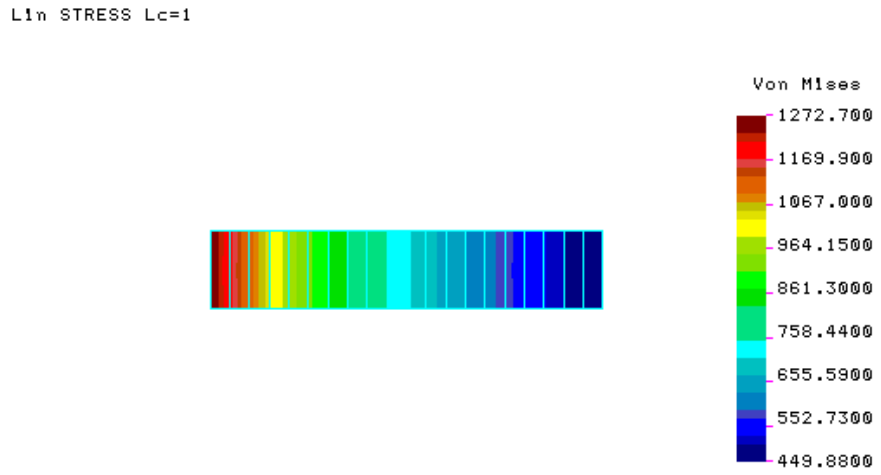
Before performing static stress analysis, you need to use the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) and **A_STRESS** (Analysis > STATIC > **Stress Analysis Options**) commands if the results from electromagnetic analysis need to be preserved in the output file:

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
Displacement Print Interval > **1**
... ...
Output Flag > **Append**

Geo Panel: Analysis > STATIC > **Stress Analysis Options**
AISC code check flag > 0
Stress print flag > **0**
... Accept Defaults ...

You can now proceed with static stress analysis using the command **R_STATIC** (Analysis > STATIC > **Run Static Analysis**). You can activate the required component of stress using **ACTSTR** command and plot the stress contours using **STRPLOT** (Results > PLOT > **Stress**). The figure below shows the von Mises stress contours.

Figure 12-8. Von Mises Stress Contours of the Solenoid Model



For a thick long solenoid, the exact solutions for circumferential and radial stresses are as follows (Moon, 1984):

$$\sigma_\theta = -\mu_0 J_0^2 b^2 \left| A - B\frac{1}{r^2} - \left(1 - \frac{3\gamma}{4}\right)\frac{\alpha^2 r^2}{2} + \left(1 - \frac{2\gamma}{3}\right)\alpha r \right|$$

$$\sigma_r = -\mu_0 J_0^2 b^2 \left| A + B\frac{1}{r^2} - \left(1 - \frac{\gamma}{4}\right)\frac{\alpha^2 r^2}{2} + \left(1 - \frac{\gamma}{3}\right)\alpha r \right| \qquad (12\text{-}2)$$

where the constants A and B are computed using the following equations:

$$A = \frac{1}{2}\left(1-\frac{\gamma}{4}\right)\left(1+\alpha^2\right)-\left(1-\frac{\gamma}{3}\right)\left(1+\alpha+\alpha^2\right)\left(1+\alpha\right)^{-1}$$
$$B = -\frac{1}{2}\left(1-\frac{\gamma}{4}\right)+\left(1-\frac{\gamma}{3}\right)\left(1+\alpha\right)^{-1} \quad (12\text{-}3)$$

In the above equations, a and b are the inner and outer diameters respectively, α=a/b, μo is the free space permeability (of air in this case), Jo is the uniform current density, and γ is defined by:

$$\gamma = (1 - 2\nu) \ / (1 - \nu)$$

Using the above equations, the circumferential stress was computed at r = 1.3 and compared against COSMOS/M results. The table below shows the results of this comparison.

Stress Component	Exact (Dyne/cm^2)	COSMOS/M (Dyne/cm^2)
σ_θ at r = 1.3	974.07	974.957

The COSMOS/M result was obtained from the output file as follows. The coordinate r = 1.3 corresponds to nodes 25 and 46. From the output file (jobname.OUT), the stress results for element 14 which connects these nodes were located. When the PLANE2D element is used in the axisymmetric mode, the circumferential component of stress is printed under SIGMA-Z header. The result indicated in the table above was therefore read from the output file as shown below:

```
 ELEMENT  STRESS  OUTPUT
 NUMBER    NODE SIGMA-X  SIGMA-Y       TAU-XY        SIGMA-Z       VON MISES     SIGMA-P+      SIGMA-P-    ANGLE

    14
          24   -.390934E+01 0.339922E+03 0.128491E-05 0.103397E+04 0.915733E+03 0.339922E+03 -.390934E+01  90.000
          25   -.390934E+01 0.320446E+03 0.128491E-05 0.974957E+03 0.863647E+03 0.320446E+03 -.390934E+01  90.000
          46   -.390935E+01 0.320446E+03 0.128491E-05 0.974957E+03 0.863647E+03 0.320446E+03 -.390935E+01  90.000
          45   -.390935E+01 0.339922E+03 0.128491E-05 0.103397E+04 0.915733E+03 0.339922E+03 -.390935E+01  90.000
      CENTER   -.390935E+01 0.330184E+03 0.128491E-05 0.100447E+04 0.889690E+03 0.330184E+03 -.390935E+01  90.000
```




13 Non-Axisymmetric Loading and Mode Shapes

Introduction

Axially symmetric models with asymmetric loading are usually modeled as three dimensional structures. However, a three dimensional problem is often expensive to **setup** and **run**. That is why axisymmetric elements with asymmetric loading options are used. A Fourier series method is used to express the given load as the sum of several harmonic component loadings and analysis is done for each load component only on a 2D axisymmetric section of the structure, thus reducing the size of the problem by several orders of magnitude. According to the principle of superposition, the original problem is solved by superposing the solutions of the component problems. Thus, the original 3D problem is replaced by a series of 2D problems.

List of Examples
Static Analysis with Asymmetric Loading.
Example 1: Static Analysis of a Circular Vessel.
Example 2: Static Analysis of a Circular Vessel.
Example 3: Static Analysis of A Nozzle.
Example 4: Static Analysis of a Nozzle.
Non-axisymmetric Mode Shape (Frequency) Analysis.
Example 5: Frequency Analysis of a Circular Vessel.
Non-axisymmetric Buckling Analysis.



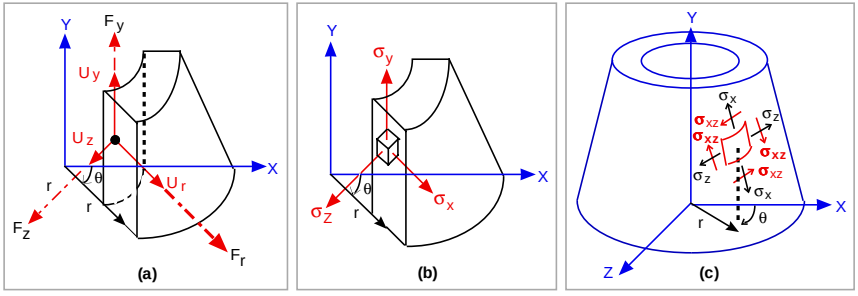

Available Models and Capabilities

There are basically two types of elements available in the STAR module of for this purpose – the PLANE2D and SHELLAX elements. In a finite element model of a solid of revolution (PLANE2D) or a thin-walled shell of revolution (SHELLAX), nodes represent circles rather than points. The Shell element is a 2-noded element whereas the axisymmetric solid has 4 to 8 nodes. The analysis is confined to linear elastic models for both isotropic and orthotropic materials. In addition to static analysis, mode shape calculation (frequency analysis) at different harmonics with or without geometric stiffening is also available for both types of elements. For buckling analysis, the program marches from the specified minimum harmonic to the maximum harmonic to find the minimum buckling load within the specified range of harmonics. The effect of gravity and/or centrifugal loading may be considered only in the axial direction.

Geometric Conventions

In general, displacements and stresses are defined in the global cylindrical coordinates according to Figure 13-1. For displacements, u_x corresponds to the radial direction, u_y corresponds to the axial direction, and u_z corresponds to the circumferential direction.

Figure 13-1.



The results are written to the output file. To display the results graphically, follow the following conventions:

1. Displacements:

   The displacements are available in the radial (u_x), axial (u_y), and circumferential (u_z) directions for the SHELLAX and PLANE2D elements as shown in Figure 13-1(a). UX, UY, and UZ are used in GEOSTAR instead of u_x, u_y, and u_z respectively.

2. Stresses:

   a. For PLANE2D elements, normal stresses are available in the radial (σ_x), circumferential (σ_y) and axial (σ_z) directions. Shear stresses are defined accordingly. Figure 13-1(b) illustrates these components. This convention is valid only when the stresses are requested in the global coordinate system as specified in the **EGROUP** command (prior to the execution of the program). For stresses requested in the local or element coordinate systems, the element cube shown in Figure 13-1 (b) will be rotated about the circumferential direction accordingly. SX, SY, and SZ are used in GEOSTAR instead of σ_x, σ_y, and σ_z respectively.

   b. For SHELLAX elements, the stresses are defined in the element coordinate system as shown in Figure 13-1(c). These directions differ substantially from the directions used for PLANE2D elements.

✍ This difference in the directions of stresses between these two types of elements should be considered when you compare the results generated from the two element types for the same model.

For more detailed description of these results as well as the calculated internal forces and moments for SHELLAX elements, please refer to the Element Library in the COSMOS/M User's Guide.

✍ For SHELLAX elements, the stress results are available for both top and bottom faces of shell element. You may use the **SHADE** (Display, Display_Options, Shaded Element Plot) command to distinguish the top face (red color by default) from the bottom face (blue color by default).

Load Input

The nodal forces defined in the X-Y plane cross section are considered to be distributed on the actual 3D model along a circular path (or surface for pressure loads). The amplitude of these loads vary in the circular direction as specified by the associated curve.

There are basically two options to define the axisymmetric loads (force/pressure) as follows:

1. **Direct Load Input:** The user defines the loads exactly as they vary along the nodal circle (in circumferential direction). When the program is executed, first it finds the equivalent Fourier series representing the load, and then proceeds to find the response for each harmonic and at the end superposes all the harmonic responses to find the actual response. This is the easier and often the more convenient of the two types of input available for axisymmetric loading.

   Notes:

   a. Only those load harmonics within the requested range (specified by the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command) will be processed. Usually, a few lower harmonics adequately approximate most load patterns. More harmonics may be needed to represent loads that vary significantly within a small circumferential angle.

   b. The equivalent Fourier series of the direct load input is available in the output file if it is requested by the input flag in the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command. Refer to the examples at the end of this chapter.

2. **Fourier Series Input:** This option may be used when the information for load vector is in this form or when the load is available essentially applied in the harmonic form. Instead of defining the actual load versus circumferential angle (direct input), the user may define up to 1000 Cosine and/or Sine coefficients of Fourier series. These coefficients are defined for load "F" according to:

$$F(\theta) = P_0 + P_1 \cos\theta + P_2 \cos 2\theta + P_3 \cos 3\theta + P_4 \cos 4\theta + \ldots + q_1 \sin\theta + q_2 \sin 2\theta + q_3 \sin 3\theta + q_4 \sin 4\theta + \ldots$$

   Thus, the response of an axially symmetric body to asymmetric loads can be analyzed by superposing component analyses, each of which represents the response due to one *component* of the total load. The Sine terms in the Fourier

series are called odd or antisymmetric, as they correspond to antisymmetric types of loads. Cosine terms are called *even or symmetric* as they represent symmetrical types of loads, i.e.,

For COSINE: $F(\theta) = F(-\theta)$

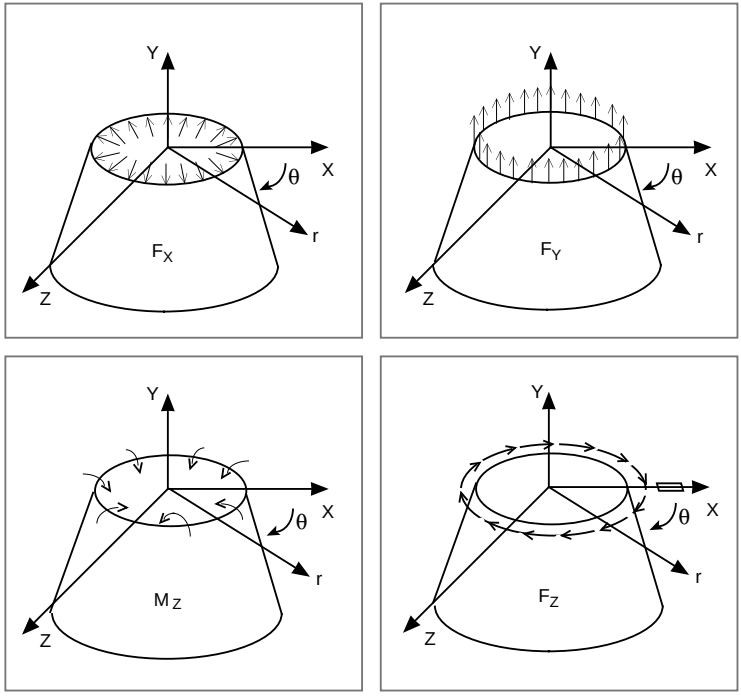
For SINE: $F(\theta) = -F(-\theta)$

Therefore, the user may consider only Cosine terms if the load is symmetric relative to the X-Y plane or Sine terms if it is distributed antisymmetrically or a combination of both for general types of loads. The following figures illustrate the loads represented by various harmonics.

Loads Corresponding to Harmonic 0 (P_o term)

This is the case of axially symmetric:

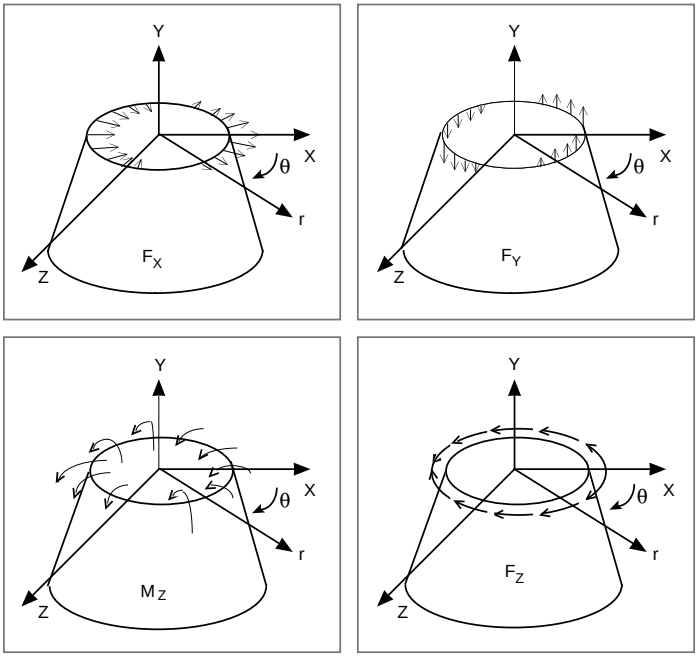
Figure 13-2.



Loads Corresponding to Harmonic 1

For Cos θ:

Figure 13-3. The Peak Values Occur at θ = 0



For Sin θ:

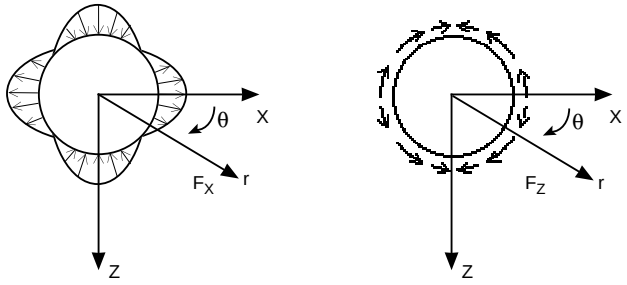
The above configuration corresponds to Sin θ, if the loads are rotated +90° in the circumferential direction θ.

Loads Corresponding to Harmonic 2

For a better visualization of this type of loading, the X-Z plane view of the loading is considered.

For Cos 2θ:

Figure 13-4.



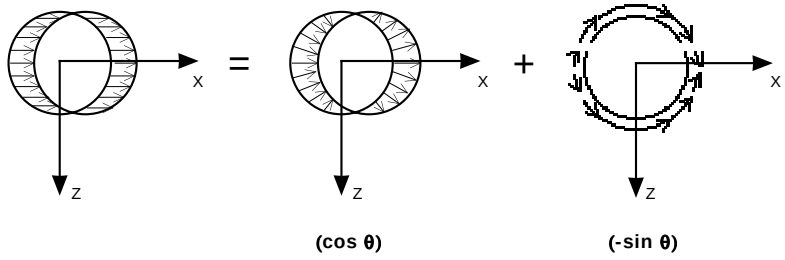
For Sin 2θ:

The above configuration corresponds to Sin 2θ, if the loads are rotated 45° in the circumferential direction θ.

Loads in Other Directions

The fundamental loads as displayed in the above figures are in radial, tangential, or axial directions. Loads in other directions may be considered by combination of loads in these directions. For example, a uniform load in the x-direction can be constructed by adding the following two harmonics.

Figure 13-5.



Command Summary and Description

The following commands are essential for asymmetric loading analysis. They provide information on the analysis type, Fourier Sine and Cosine coefficients, and asymmetric loads defined as a function of circumferential angle, θ.

A_ASYM (Analysis > STATIC > **Asymmetric Load Options**)

ACTSET (Control > ACTIVATE > **Set Entity**)

CURDEF (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**)

CURLIST (LoadsBC > FUNCTION CURVE > **List Time/Temp**)

CURDEL (LoadsBC > FUNCTION CURVE > **Delete Time/Temp Curve**)

In the following, brief outlines involving the application of specific commands required for different analysis options are presented.

Static Analysis with Asymmetric Loading

In the following, only the commands essential to set up the static analysis of an axisymmetric model under asymmetric loading will be listed. The model is assumed to have been defined.

1. **EGROUP** Command:
   a. For the PLANE2D element, the asymmetric option must be assigned to activate the Z-coordinate as an additional Degree of Freedom that will be considered in the formulation of this element.
   b. For the SHELLAX element, no special option is needed to be activated.

2. **ANALYSIS** Command:

   The **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command specifies that the loading (or mode shapes, in case of Frequency or Buckling) will be non-axisymmetric and also it will specify whether the load is being applied as an odd or even Fourier expansion and, finally, the circumferential angles at which output results for the printing and plotting are desired. This command should be issued prior to the **ACTSET** and **CURDEF** commands, as noted below. Use the **"A_LIST, STATIC"** (Analysis > **List Analysis Option**) command to list flags for static analysis, including flags related to asymmetric loading.

3. **ACTSET** Command:

   This command activates the type of load and the pattern set number. By assigning FCOS, FSIN, or FCOEF to the first prompt, the active load type will be defined in the form of a Fourier series type (Cosine and/or Sine) or direct load input (load versus circumferential angle). The second prompt corresponds to the pattern set number. The user can define up to 25 patterns for a Fourier series type and/or 25 patterns for direct load input. The load will be assigned to the active pattern set number (see the following example).

4. **CURDEF** Command:

   Defines the load versus circumferential angle or the coefficients of Fourier series for different harmonics. These load coefficients are considered as multipliers to the actual loads defined by GEOSTAR "force" or "pressure" commands. Use the **CURLIST** (LoadsBC > FUNCTION CURVE > **List Time/Temp**) command to list the harmonics assigned or use the **CURDEL** (LoadsBC > FUNCTION CURVE > **Delete Time/Temp Curve**) command to delete any




curve. Note that only the range of harmonics assigned in the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) will be considered in the computations. The assigned out-of-range coefficients of the **CURDEF** (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**) command will be ignored. (You may use the option for reading the curve values from external files.)

For direct load input (load versus circumferential angle, θ), the range of angle θ is $-180° < \theta < 180°$ for general option (Sine + Cosine terms). However, if the load has even or odd characteristics and the Sine option or Cosine option is intended to be used, the range of the angle θ (for which the load is to be defined) reduces to $0° < \theta < 180°$.

5. LISTING Commands:

   In addition to the **A_LIST** (Analysis > **List Analysis Option**) and **CURLIST** (LoadsBC > FUNCTION CURVE > **List Time/Temp**) commands already mentioned in items 2 and 4, you may list the defined forces or pressures by using the **FLIST** (LoadsBC > STRUCTURAL > FORCE > **List**) or the **PLIST** (LoadsBC > STRUCTURAL > PRESSURE > **List**) commands. The first listed row for each node or element corresponds to the force or pressure value and the second row corresponds to the associated curve number.

6. **PRINT_OPS** Command:

   Activating the "detailed print option" flag of this command provides (in addition to the regular output) the maximum displacements and stresses of each individual harmonic in the output file. These maximum correspond to the maximum response of the structure along each circumferential circle.

7. Postprocessing Commands:

   a. Cross-sectional Plots

      The displacements and stresses for each one of the circumferential angles, assigned by the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command, are stored as a load case. Use the **RESULTS** (Results > **Available Results**) command to identify the available load cases, and then use the **ACTDISP** or **ACTSTR** command to activate the desired load case (i.e. circumferential angle). To display the results, use the **DISPLOT** or **STRPLOT** (Results > PLOT > **Displacement, Stress**) commands. In addition, you may use the "list" commands, such as **DISPLIST** or **STRLIST** (Results > LIST > **Displacement, Stress Component**) to list the results.

b. 3D Plots

Displacement, deformation, and stress plots may be plotted over a 3D display of the model. Results are displayed in the cylindrical coordinate system according to coordinate directions defined in Figure 13-1. To be able to generate 3D plots, you must activate the *special element plot* flag in the **SETEPLOT** (Meshing > ELEMENTS > **Set Element Plot**) command and activate shading using the **SHADE** (Display > DISPLAY OPTION > **Shaded Element Plot**) command. The **PSCALE** (or Perfect Scaling icon) command should also be used for scaling. With this setting, all plots will be 3D based.

The following examples illustrate the use of commands pertinent to the asymmetric loading. Additional examples are included in the verification problem section of the manual (problems S79 through S83).




Example 1: Static Analysis of a Circular Vessel

Line forces defined by the "Direct Load Input" option.

$F_R(\theta) = F_R(-\theta)$			
q°	F_R^1	F_R^2	FR^3
0	–20000	–10000	–5000
3	–19000	–9500	–4400
6	–17800	–8900	–3700
9	–16400	–8200	–2900
12	–14600	–7400	–2000
15	–13000	–6500	–1000
18	–11000	–5500	0
21	–8800	–4400	0
24	–6400	–3200	0
27	–3800	–1900	0
30	–1000	–500	0
31	0	0	0

Given:

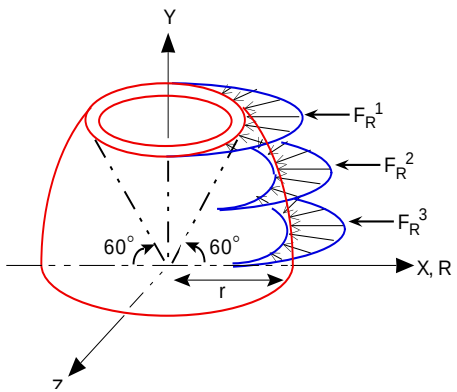
E = 30,000 ksi

r = 10 in

Thickness = 0.1

Poisson Ratio = 0.3

Figure 13-6.



Loads:

Loads are applied in the radial direction and are symmetric relative to the X–Y plane with their variations shown in the above figure. Radial loads F_R^1 and F_R^2 distributed along the circumference have the same pattern and the ratio of their amplitudes is fixed for different angles and, as will be shown later, they may be represented with the same pattern set. However, F_R^3 has a different pattern from the other two loads and must be assigned to another pattern set. Only the loads in the positive θ have to be defined. Note that the loads in the

figure are not drawn to scale. Use F_x in place of radial forces (F_R) to model the problem.

GEOSTAR Commands

The following commands cover the modeling of the problem.

Geo Panel: Propsets > **Element Group**
EGROUP,1,SHELLAX,0,0,0,0,0,0,0,

(Defining the element group as SHELL)

Geo Panel: Propsets > **Material Property**
MPROP,1,EX,3.E7,NUXY,.3,

(Defining the material properties)

Geo Panel: Propsets > **Real Constant**
RCONST,1,1,1,1,0.1,

(Defining the thickness of SHELL)

Geo Panel: Geometry > GRID > **Plane**
PLANE,Z,0,1,

Use View icon for X-Y
VIEW,0,0,1,0,

Geo Panel: Geometry > POINTS > **Define**
PT,1,0,0,0,
PT,2,10,0,0,

Geo Panel: Geometry > CURVES > CIRCLES > **Circle in Plane**
CRPCIRCLE,1,1,2,10,60.,1,

(Defining the circular geometry of SHELL)

Geo Panel: Meshing > PARAMETRIC MESH > **Curves**
M_CR,1,1,1,2,30,1,

(Generating 30 elements)

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
DND,1,ALL,0,1,1,
DND,2,RY,0,31,1,RX,,

(Apply the displacement constraints)

Use Auto Scaling icon
SCALE,0,

Commands Pertinent to Asymmetric Loading

Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option >**Yes**

(Activating the asymmetric type)

Harmonic type > **COS**

(Considering only Cosine harmonic)
Beginning Harmonic number > **0**

(Start from harmonic 0)
Ending Harmonic number > **10**

(End at harmonic 10)
Starting angle for output > **0**

(Start at 0 circumferential angle for output results)
Ending angle for output > **360**

(End at angle 360°)
Increment angle for output > **10**

(At intervals of 10°)

You may use the command **"A_LIST, STATIC"** (Analysis > **List Analysis Option**) to verify your input. In the above, we utilized the symmetrical pattern of loads relative to circumferential direction and assigned "Cosine" to avoid unnecessary computation for non-existent (or numerically negligible) "Sine" harmonics. However, if there is any doubt about the distribution of the load, you may use the general option (both Sine and Cosine terms) provided that you define the loading along the entire circular path. The number of angles for result output should not exceed 50.

Now, we define the first pattern set for loads F_R^1 and F_R^2:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **FCOEF: Fourier Cosine Coeff**

(Assign the Input Load Option)
Coeff. set > **1**

(Pattern set number 1)

Now, we define the load curve pattern:

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOEF: Fourier Cosine Coeff**
(Assign "Direct Load Input")
Curve number > **1**
(Pattern set 1)
Start point > **1**
(Start from beginning)
Circumf. angle for point 1 > **0**
Circumf. load for point 1 > **20000**
.
.
.
Circumf. angle for point 10 > **27**
(Define the first 10 points on the curve)
Circumf. load for point 10 > **3800**

Since **CURDEF** only allows 10 points to be defined every time it is used, you have to repeat the command to define points 11 and 12.

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type >FCOEF; Fourier Cosine Coeff
Curve number > **1**
(For the same set No. 1)
Start point > **11**
(Start from point 11)
Circumf. angle for point 11 > **30**
Circumf. load for point 11 > **1000**
(Define the last 2 points)
Circumf. angle for point 12 > **31**
Circumf. load for point 12 > **0**
(Refer to the on-line help to read curve values from external files.)

You may use the commands **"CURLIST, FCOEF"** to verify your input.

Next, define the radial loads F_R^1 and F_R^2 with the proper load multipliers –1 and –0.5 to account for the negative sign of the load and the differences in the magnitudes of the two.

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **31**
Force label > **FX: X Force**
Value > **-1**
Ending node > **31**
Increment > **1**

(Define the load curve multiplier for F_R^1 at node 31)

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **20**
Force label > **FX: X Force**
Value > **-0.5**
Ending node > **20**
Increment > **1**

(Define the load curve multiplier for F_R^2 at node 20)

(Note that F_x stands for radial loads in the command structural)

You may use **FLIST** (LoadsBC > STRUCTURAL > FORCE > **List**) command to verify the latest input.

Now, we have to activate set number two for the third load (F_R^3).

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **FCOEF: Fourier Cosine Coeff**

Coefficient set > **2**

(Activate set No. 2)

Defining the load pattern:

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOEF: Fourier Cosine Coeff**
Curve number > **2**

(Define the second curve)
Start point > **1**
Circumf. angle for point 1 > **0**
Circumf. load for point 2 > **5000**
.
.
.

Circumf. angle for point 7 > **18**
Circumf. load for point 7 > **0**

Defining the load multiplier for F_R^3:

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
Beginning node > **10**
Force label > **FX: X Force**
Value > **-1**
Ending node > **10**
Increment > **1**

(Assigns value of –1)

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
Displacement print flag > **Yes**
... ...
Output flag > **Append**

(Requesting maximum response at each harmonic)

Geo Panel: Analysis > STATIC > **Run Static Analysis**

(Evaluate the displacements and stresses and print the results)

Results

1. For displacements and stresses, the output printout contains not only the total response at the requested circumferential angles, but also the maximum response (displacement and stresses) along the circumferential circle for each individual harmonic if it has been requested by the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command (see the modeling section of this example).

2. In addition to the maximum response along the circumferential circle for each node, the maximum among all the nodes is also available in the printout for each displacement harmonic as well as for the total displacement.

3. The individual harmonic response helps in finding out the contribution of each harmonic to the total response. Therefore, it is a good indicator for the termination of higher non-essential harmonics in subsequent runs, resulting in less computational efforts.




4. Bearing in mind the above statement, one may first run the displacement analysis only (by suppressing the stress calculation flag in the **STRESS** (Analysis > STATIC > **Activate Stress Calc**) command, and after reviewing the contribution of each harmonic to the total displacement, select a smaller range of harmonics (by reissuing the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command) and run stresses for the new reduced range.

5. You may run stresses for a different range of angles without having to recalculate displacements.

6. Displacements and stresses are in element coordinates (see the *Geometric Convention* section of this chapter).

7. You may display the results by using the "display" commands. The results for each circumferential angle, specified by the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command, is stored in the plot file as a load case.

8. Results may be displayed for the full 3D model as shown below.

9. Use the **SHADE** command prior to plotting the elements in 3D view for SHELLAX element to distinguish the top (red color) surface from the bottom surface (blue color). This helps to distinguish the stresses on the two surfaces of element.

Postprocessing Using 3D Model

Postprocessing may be performed on a cross-section or on the 3D model. Cross-section postprocessing is useful in the case of area elements like PLANE2D and TRIANG. Postprocessing SHELLAX elements should be processed using the 3D model. To activate 3D postprocessing, activate the special element plotting flag in the **SETEPLOT** (Meshing > ELEMENTS > **Set Element Plot**) command and activate shading as follows.

Geo Panel: Meshing > ELEMENTS > **Set Element Plot**
Element plot order > **Linear**
Plot beam orientation > **No**
Special element plot > **Yes**

Geo Panel: Display > DISPLAY OPTION > **Shaded Element Plot**

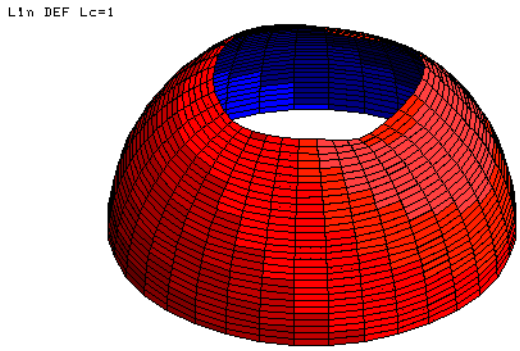
Deformation Plot

Click on View icon, isometric

Geo Panel: Results > PLOT > **Deformed Shape**

Click on Auto Scaling icon

Figure 13-7. 3D Deformation Plot



A resolution of 10 degrees in the circumferential direction is used as specified by the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command prior to running analysis. Refer to figure #1 in Figure 13-1 for the positive direction of the circumferential angle.

Displacement Contour Plot

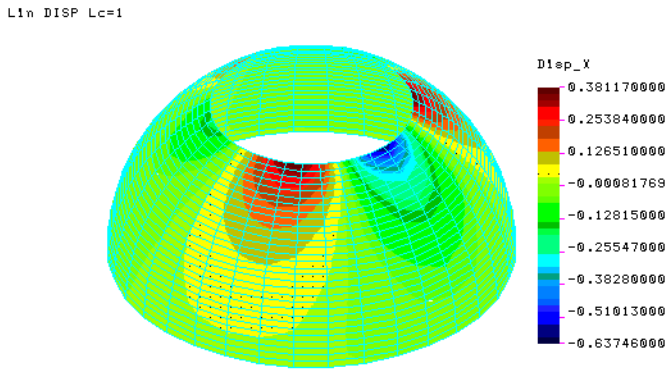
Contour plots are available for plotting at the resolution specified in the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command. The following commands will generate a contour plot for the displacement in the radial direction. Note that **SETEPLOT** (Meshing > ELEMENTS > **Set Element Plot**) and **SHADE** (Display > DISPLAY OPTION > **Shaded Element Plot**) commands must have been issued.

Geo Panel: Results > PLOT > **Displacement**

(For UX in the deformed configuration)

Click on the Auto Scaling icon

Figure 13-8. Radial Displacement Contour Plot



Stress Plot

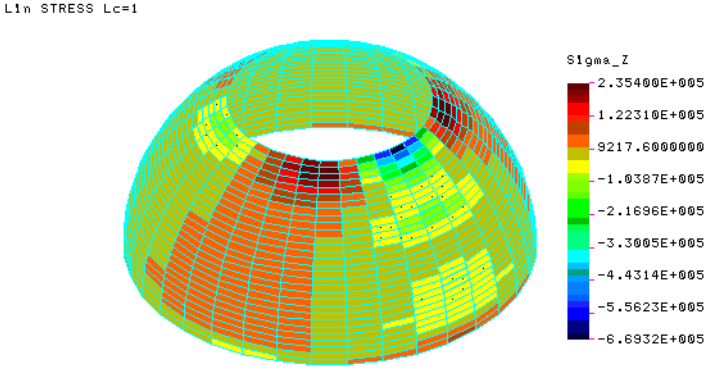
Stress results for SHELLAX elements are available in the local element coordinate system as specified in Figure 13-1. The following commands generate the hoop stress contour plot. Note that **SETEPLOT** and **SHADE** must have been issued earlier.

Geo Panel: Results > PLOT > **Stress**

(For SZ element stresses in the deformed configuration)

Click on Auto Scaling icon

Figure 13-9. Hoop Stress Contour Plot



Stress results for SHELLAX elements are only available in the local element coordinate system. Only element stresses are available. The stress values displayed between two consecutive angles in the tangential direction correspond to the values at the beginning of the interval.

Improving Resolution

You may improve the displayed results by specifying a smaller angle resolution at any time and run the stresses without having to run displacements. You may for example recalculate the stresses with better resolution between angles 270 and 360 degrees as follows:

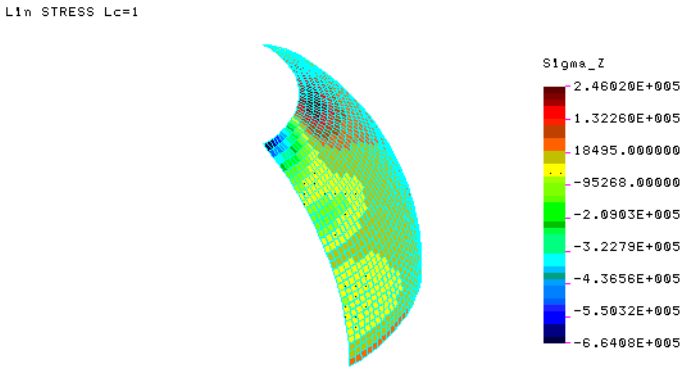
Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option > **Yes**

Harmonic type > **COS**
Beginning Harmonic number > **0**
Ending Harmonic number > **10**
Starting angle for output > **270**
Ending angle for output > **360**
Increment angle for output > **2**

Geo Panel: Analysis > STATIC > **Run Stress Analysis**

Figure 13-10 shows the improved Hoop stress contour plot.

Figure 13-10. Improved Hoop Stress Plot



Note that when the stress angle resolution is different than that of displacements, then you may not plot stresses on the deformed shape.

Example 2: Static Analysis of a Circular Vessel

Pressure loads defined by the “Direct Load Input” option.

q°	P
–150	500
–135	1100
–120	1800
–105	2600
–90	3400
–75	4200
–60	5000
–45	5000
–30	4100
–15	3250
0	2450
+15	1700
+30	1000
+45	350
+55	0

Given:

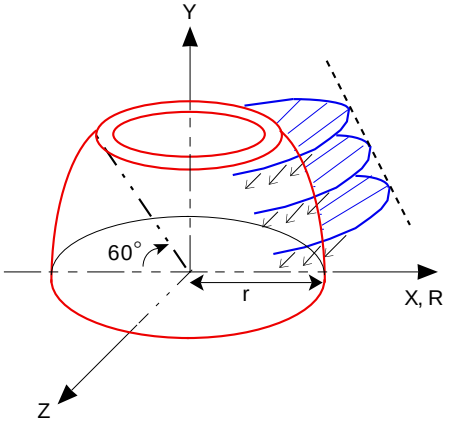
E = 30,000 ksi

r = 10 in

Thickness = 0.1

Poisson Ratio = 0.3

Figure 13-11.



Loads:

Loads are applied normal to the outer surface of the shell, and are defined in the above figure (not drawn to scale). We only need to define one load pattern set for the problem. For problems with more than one load pattern, the procedure is the same as described in the previous example.

Contrary to the previous problem, due to the lack of symmetry (in the circumferential direction), the load along the entire circular path must be defined.

GEOSTAR Commands

Please refer to previous example for descriptions of the commands below.

Geo Panel: Propsets > **Element Group**
EGROUP,1,SHELLAX,0,0,0,0,0,0,0,

Geo Panel: Propsets > **Material Property**
MPROP,1,EX,3.E7,NUXY,.3,

Geo Panel: Propsets > **Real Constant**
RCONST,1,1,1,1,0.1,

Geo Panel: Geometry > GRID > **Plane**
PLANE,Z,0,1,

Use View icon for X-Y view
VIEW,0,0,1,0,

Geo Panel: Geometry > POINTS > **Define**
PT,1,0,0,0,
PT,2,10,0,0,

Geo Panel: Geometry > CURVES > CIRCLES > **Circle in Plane**
CRPCIRCLE,1,1,2,10,60.,1,

Geo Panel: Meshing > PARAMETRIC MESH > **Curves**
M_CR,1,1,1,2,30,1,

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
DND,1,ALL,0,1,1,
DND,2,RY,0,31,1,RX,,

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
PRINT_OPS,1,0,0,1,0,1,0,0,0,1,

Use Auto Scaling icon
SCALE,0,

Commands Pertinent to Asymmetric Loading

Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option > **Yes**

(Activating the asymmetric type)

Harmonic type > **GEN**

(Considering both Sine and Cosine terms)

Beginning Harmonic number > **0**

(Start from harmonic 0)
Ending Harmonic number >**10**

(End at harmonic 10)
Starting angle for output > **0**

(Start at 0 circumferential angle)
Ending angle for output > **360**

(End at angle 360°)
Increment angle for output > **90**

(At intervals of 90°)

In the above command, we have specified that the analysis be performed by considering both Sine and Cosine harmonic terms (general option). It is also assumed that 10 harmonics will be sufficient although, at the end, the user can examine the contributions from each harmonic to determine if more or less are required for subsequent runs as noted in the previous example.

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **FCOEF: Fourier Cosine Coeff**

(Assign the Input Load Option)

Coeff. set > **1**

(Assign the load set)

Defining the load curve pattern:

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOEF: Fourier Cosine Coeff**

Curve number > **1**

Start point > **1**

Circumf. angle for point 1 > **–150**
Circumf. load for point 1 > **500**
.
.
.
Circumf. angle for point 10 > **–15**
Circumf. load for point 10 > **3250**

(Define the first 10 points)

Defining the rest of the curve:

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOEF: Fourier Cosine Coeff**

Curve number > **1**

Start point > **11**

Circumf. angle for point 11 > **0**
Circumf. load for point 11 > **2450**
.
.
.
Circumf. angle for point 15 > **55**
Circumf. load for point 15 > **0**

(Define the last 5 points)

Defining the pressure on the elements at the tip:

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > **Define Element**
PEL,29,–50, 1,30,1,4

(Pressure defined on elements 29 and 30)

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
PRINT_OPS,1,0,0,1,0,1,0,0,1,0,

Geo Panel: Analysis > STATIC > **Run Static Analysis**
R_STATIC

(Evaluate the response and print the results)

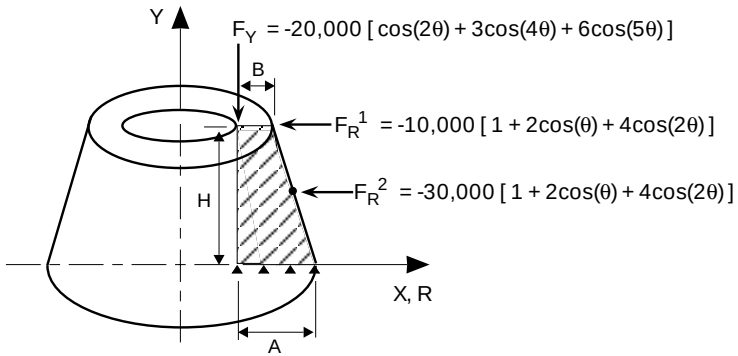
For options available for the postprocessing, please refer to the previous example.




Example 3: Static Analysis of a Nozzle

Loads defined by “COSINE” Fourier Series.

Figure 13-12.



Given:

E	= 30,000 ksi
H	= 30 in.
A	= 10 in.
B	= 5 in.
Poisson Ratio	= 0.3

Loads:

Loads are distributed around the tip of the nozzle and are approximated by Fourier series as shown in the above figure. The two radial forces (FR) have the same pattern (though different amplitudes), therefore only one pattern is sufficient to define both of them (Set 1 in the following commands). However, for axial load (FY) it is necessary to define a new pattern (Set 2).

GEOSTAR Commands

The following commands cover the modeling of the problem.

Geo Panel: Geometry > GRID > **Plane**
PLANE,Z,0,1,

(Project the X–Y Plane)

Use View icon to set X-Y view
VIEW,0,0,1,0,

Geo Panel: Geometry > CURVES > **Draw Polyline**
CRSPOLY,1,10,0.,0.,L,10.,30.,0.,L,15.,30.,0.,L,20,0.,0.,L,10.,0.,0.,

Use Scaling icon
SCALE,0,

(Form the boundary of the section)

Geo Panel: Geometry > SURFACES > **Define by 2 Cr**
SF2CR,1,2,4,0,

(Assign a surface by 2 opposite edges)

Geo Panel: Propsets > **Element Group**
EGROUP,1,PLANE2D,0,0,3,0,0,0,0,

(Define a PLANE2D element with asymmetric option)

Geo Panel: Propsets > **Material Property**
MPROP,1,EX,30E6,NUXY,.3,

(Define Material Properties)

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
M_SF,1,1,1,4,5,8,1,1,

(Generate 4-node elements)

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
DCR,4,ALL,0,4,1

(Constrain bottom end of the nozzle)

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
PRINT_OPS,1,0,0,1,0,1,0,0,1,0,

(Ask for detailed printout)

In the above **EGROUP** (Propsets > **Element Group**) command, it is essential to activate the asymmetric option for the PLANE2D element (not required for the SHELLAX element).

Commands Pertinent to Asymmetric Loading

Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option > **Yes**

(Activating the asymmetric type)

Harmonic type > **COS**

(Considering only Cosine harmonic)

Beginning Harmonic number > **0**

(Start from harmonic 0)
Ending Harmonic number > **5**

(End at harmonic 5)
Starting angle for output > **0**

(Start at zero circumferential angle for output results)
Ending angle for output > **90**

(End at angle 90°)
Increment angle for output > **10**

(At intervals of 10°)

You may use the command **"A_LIST, STATIC"** (Analysis > **List Analysis Option**) to check your input for the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command.

Now, we define the first pattern set for loads F_R^1 and F_R^2:

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **FCOS: Fourier Cosine Coeff**

COSINE Coeff. set > **1**

(Activate COSINE Fourier Series set No. 1 for loads)

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOS: Fourier Cosine Coeff**

(For COSINE Fourier)

Curve number > **1**

(Set Pattern Number 1)

Start point > **1**

(Start from the beginning)

Cosine Harmonic for point 1 > **0**

(For Harmonic 0)

Fourier Cosine Coeff. for point 1 > **1**

(Define 1 as the coefficient of Harmonic)

Cosine Harmonic for point 2 > **1**

Fourier Cosine Coeff. for point 2 > **2**

(For Harmonic 1, assign 2 as coeff.)

Cosine Harmonic for point 3 > **2**

Fourier Cosine Coeff. for point 3 > **4**

(For Harmonic 2, assign 4 as coeff.)

The above command defines pattern 1 for both radial loads (as was mentioned earlier in the "load" section of this example).

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**

FND,6,FX,–10000.,6,1

(Define a load of –10000 at node 6 in radial direction, FR, as the multiplier to the current active Set 1)

FND,30,FX,–30000.,30,1,

(Similarly define a radial load of –3000 at node 30 for the same active Set 1)

Since the axial load in this example consists of a different harmonic set, we have to define a new set:

Geo Panel: Control > ACTIVATE > **Set Entity**

Set label > **FCOS: Fourier Cosine Coeff**

COSINE Coeff. set >2

(Active Cosine Fourier set No. 2)

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**

Curve type > **FCOS: Fourier Cosine Coeff**

Curve number > **2**

(For Pattern Number 2)

Start point >1




Cosine Harmonic for point 1 > **2**
Fourier Cosine Coeff. for point 1 > **1**
Cosine Harmonic for point 2 > **4**

(Define the second set for Fourier loads)

Fourier Cosine coeff. for point 2 > **3**
Cosine Harmonic for point 3 > **5**
Fourier Cosine coeff. for point 3 > **6**

You may use the **"CURLIST, FCOS"** (LoadsBC > FUNCTION CURVE > **List Time/Temp**) command to check your input.

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**
FND,1,FY,–20000,1,1

(Define the axial load of –20000 as the multiplier to the active Fourier set No. 2 at node 1)

Geo Panel: Analysis > STATIC > **Run Static Analysis**
R_STATIC

(Evaluate the displacements and stresses and print out the results)

✍ Any harmonic in the range specified by **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**), for which there is no load factor, will have zero contribution for the total response. For instance, harmonic 3, which is absent in both load patterns, has no contribution to total response as can be verified in the output file by looking at the results for maximum displacement at harmonic 3. Also, any harmonic terms of the specified load which are not included in the range of computation (specified by the **A_ASYM** command) will be ignored.

Postprocessing Using Cross-Section Display

As mentioned earlier, you may postprocess results in 2D or 3D format. The special element plot flag in the **SETEPLOT** (Meshing > ELEMENTS > **Set Element Plot**) command controls the type of postprocessing. The default is to use 2D postprocessing. You should deactivate the special element plot flag in the **SETEPLOT** command to return to default value if you have activated it earlier. The results are available in the element coordinate system.

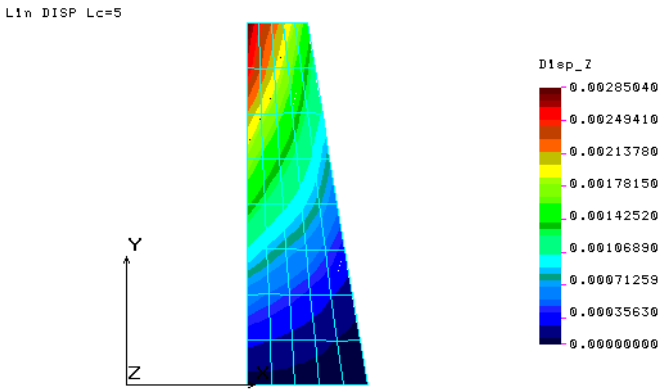
Displacement Plot

Results at the circumferential angle 40 degrees are stored under load case 5. To display displacements in the tangential direction at 40 degrees:

Geo Panel: Meshing > ELEMENTS > **Set Element Plot**
SETEPLOT;

Geo Panel: Results > PLOT > **Displacement**
ACTDIS, 5, UZ,
DISPLOT;

Figure 13-13. Tangential Displacements at 40 Degrees

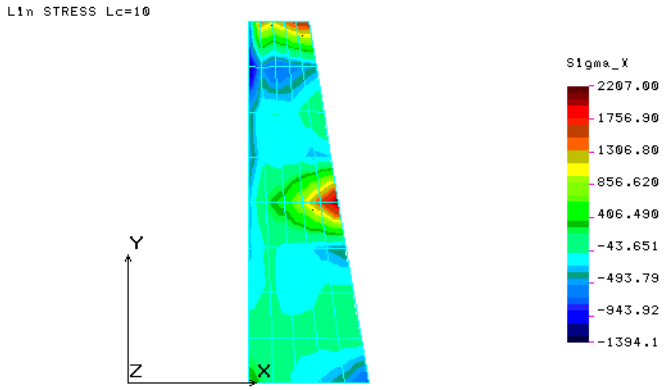


Stress Plot

Radial stresses at angle 90 (stored in load case 10) can be plotted as follows:

Geo Panel: Results > PLOT > **Stresses**
ACTSTR, 10, SX;
STRPLOT;

Figure 13-14. Radial Stresses at 90 Degrees



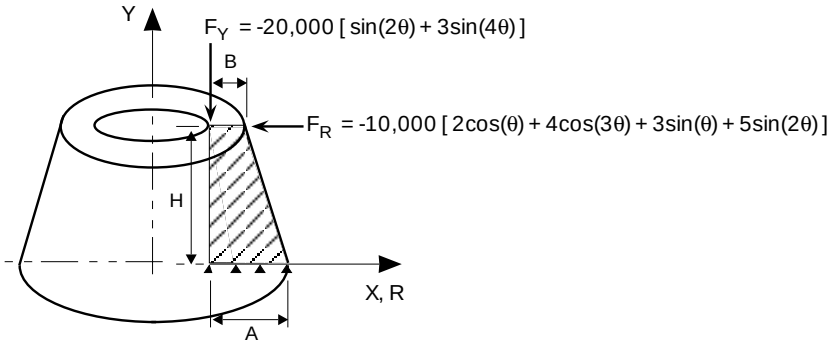
Postprocessing Using the 3D Display

To be able to view results in 3D, you need to activate the special element plotting using the **SETEPLOT** (Meshing > ELEMENTS > **Set Element Plot**) command and shading has to be activated. You may proceed with generating deformed shape plots and displacement and stress contours as has been described in the first example in this chapter.

Example 4: Static Analysis of a Nozzle

Loads defined by COSINE and/or SINE terms of Fourier Series.

Figure 13-15



Given:

E	= 30,000 ksi
H	= 30 in.
A	= 10 in.
B	= 5 in.
Poisson Ratio	= 0.3

Loads:

In this example, the radial force (F_R) consists of both Cosine and Sine terms whereas the axial load (F_Y) consists of only the Sine terms.

GEOSTAR Commands

Commands pertinent to modeling.

Geo Panel: Geometry > GRID > **Plane**
PLANE,Z,0,1,

(Project the X–Y Plane)

Use View icon to set X-Y view
VIEW,0,0,1,0,

Geo Panel: Geometry > CURVES > **Draw Polyline**
CRSPOLY,1,10,0.,0.,L,10.,30.,0.,L,15.,30.,0.,L,20,0.,0.,L,10.,0.,0.,

Use Auto Scale icon
SCALE,0,

(Form the boundary of the section)

Geo Panel: Geometry > SURFACES > **Define by 2 Cr**
SF2CR,1,2,4,0,

(Assign a surface by 2 opposite edges)

Geo Panel: Propsets > **Element Group**
EGROUP,1,PLANE2D,0,0,3,0,0,0,0,

(Define a PLANE2D element with asymmetric option)

Geo Panel: Propsets > **Material Property**
MPROP,1,EX,30E6,NUXY,.3,

(Define Material Properties)

Geo Panel: Meshing > PARAMETRIC MESH > **Surfaces**
M_SF,1,1,1,4,5,8,1,1,

(Generate 4-node elements)

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Curves**
DCR,4,ALL,0,4,1,

(Constrain bottom end of the nozzle)

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
PRINT_OPS,1,0,0,1,0,1,0,0,1,0,

(Ask for detailed printout)

As mentioned in the previous example, it is essential to activate the asymmetric condition (Option 3) for the PLANE2D element (defined by the **EGROUP** command).

Commands Pertinent to Asymmetric Loading

Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option > **Yes**

(Activating the asymmetric type)

Harmonic type > **GEN**

(Considering both Sine and Cosine harmonics)

Beginning Harmonic number > **1**

(Start from harmonic 1)

Ending Harmonic number > **4**

(End at harmonic 4)

Starting angle for output > **0**

(Start at zero circumferential angle for output results)

Ending angle for output > **360**

(End at angle 360°)

Increment angle for output > **10**

(At intervals of 10°)

In the above command, we have instructed the program to analyze from harmonic 1 to 4 (which covers the entire range of load harmonics) for both Sine and Cosine harmonics. The fact that the load FY has no Cosine term assignments causes no problems since the program internally assigns zero coefficients for missing load harmonics.

(You may use the list command "**A_LIST, STATIC**" (Analysis > **List Analysis Option**) to check your input.)

Geo Panel: Control > ACTIVATE > **Set Entity**

Set label > **FSIN: Fourier Sine Coeff**

SINE Coeff. set > **1**

(Activate SINE Fourier Series set No. 1 for axial loads)

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**

Curve type > **FSIN: Fourier Sine Coeff**

(For SINE Fourier)

Curve number > **1**

(Set Pattern Number 1)

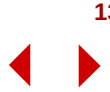
Start point > **1**

(Start from the beginning)

Sine Harmonic for point 1 > **2**

(For Sine Harmonic 2)



Fourier Sine Coeff. for point 1 > **1**

(Define 1 as the coefficient of Harmonic)

Sine Harmonic for point 2 > **4**

Fourier Sine Coeff. for point 2 >3

(For Harmonic 4, assign 3 as coefficient)

You may use the **"CURLIST, FSIN"** (LoadsBC > FUNCTION CURVE > **List Time/Temp**) command to check your input.

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**

FND,1,FY,–20000,1,1

(Define the axial load of –20000 as the multiplier to the active Fourier set No. 1 at node 1)

Now, let us define the radial load (F_R):

Geo Panel: Control > ACTIVATE > **Set Entity**

Set label > **FSIN: Fourier Sine Coeff**

SINE Coeff. set > **2**

(Activate SINE Fourier set No. 2)

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**

Curve type > **FSIN: Fourier Sine Coeff**

(For SINE Fourier)

Curve number > **2**

(Set Pattern Number 2)

Start point > **1**

(Start from the beginning)

Sine Harmonic for point 1 > **1**

(For Sine Harmonic 1)

Fourier Sine Coeff. for point 1 >

(Define 3 as the Harmonic coefficient)

Sine Harmonic for point 2 > **2**

Fourier Sine Coeff. for point 2 > **5**

(For Harmonic 2, assign 5 as coeff.)

Before issuing the force command, we have to define the Cosine part of the Fourier series by first activating the Cosine terms for the same active set (set number 2).

Geo Panel: Control > ACTIVATE > **Set Entity**
Set label > **FCOS: Fourier Cosine Coeff**

COSINE Coeff. set > **2**

(Activating the same set number for the Sine terms of radial load)

Geo Panel: LoadsBC > FUNCTION CURVE > **Time/Temp Curve**
Curve type > **FCOS: Fourier Cosine Coeff**

(For Cosine Fourier)

Curve number > **2**

(Set Pattern Number 2)

Start point > **1**

(Start from the beginning)

Cosine Harmonic for point 1 > **1**

(For Cosine harmonic 1)

Fourier Cosine Coeff. for point 1 > **2**

(Define 2. as the harmonic coeff.)

Cosine Harmonic for point 2 > **3**
Fourier Cosine coeff. for point 2 > **4**

(For harmonic 3, assign 4. as coeff.)

Geo Panel: LoadsBC > STRUCTURAL > FORCE > **Define Nodes**

(Define the radial load of –10000 as the multiplier to the active SINE and COSINE with the same set No. 2)

Geo Panel: Analysis > STATIC > **Run Static Analysis**

(Evaluate the displacements and stresses and print out the results)

For proper interpretation of the results, please refer to the notes at the end of the first example.

Use the **ACTDISP** or **ACTSTR** command in combination with the **DISPLOT** (Results > PLOT > **Displacement**) or **STRPLOT** (Results > PLOT > **Stresses**) command to display the results at certain circumferential angles.

Non-Axisymmetric Mode Shape (Frequency) Analysis

The frequencies and corresponding mode shapes can be calculated for any specified harmonic number. The program can evaluate up to 200 mode shapes of the structure for the requested harmonic depending on the eigenvalue extraction technique used. You may use Subspace, Lanczos, Power Inverse techniques for this purpose, as well as the frequency shift, Sturm Sequence check, geometric stiffening, and all other available options of the DSTAR Module of COSMOS/M. The mode shapes are evaluated at different circumferential angles along the circumferential direction and are printed in the output file. The Mode Shape evaluation is available for both PLANE2D and SHELLAX elements.

The commands essential to set up the asymmetric mode shape computation are described below. The model is assumed to have been defined.

1. **EGROUP** Command:

   Similar to the static analysis, the Asymmetric Option must be activated for the PLANE2D element (for the SHELLAX element, no additional options are required to activate).

2. ANALYSIS Command:

   a. Choice of harmonic (Sine or Cosine) is irrelevant here. Use any option.

   b. Your choice of harmonic number is specified only by the beginning harmonic prompt, therefore, the ending harmonic prompt is ineffective here (use default).

   The prompts for angles are applicable to the results in the output file for writing and plotting of the mode shapes. To account for geometric stiffening (if desired), the load must be applied only in an axisymmetric manner as is done for regular static problems (see the following example).

   The **A_FREQUENCY** (Analysis > FREQUENCY/BUCKLING **> Frequency Options**) command specifies the required parameters for eigenvalue solution such as the eigenvalue technique, number of mode shapes, frequency shift, in-plane stiffening effects, and so on. However, the harmonic number input is ignored in this case.

Stresses can also be evaluated for all mode shapes by issuing the **R_STRESS** (Analysis > STATIC > **Run Stress Analysis**) command and are available in the output file as well as in the plot file for maximum values along the circumferential direction.

3. LISTING Commands:

   Use the **"A_LIST, FREQ"** (Analysis > **List Analysis Option**) command to list general assignments for frequency analysis, including assignments made by the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command. Use the **FLIST** (LoadsBC > STRUCTURAL > FORCE > **List**) or **PLIST** (LoadsBC > STRUCTURAL > PRESSURE > **List**) command to list applied loads in case the geometric stiffening effects are considered.

4. **PRINT_OPS** Command:

   Activating the "detailed print option" flag of this command (with the mode shape flag of **A_FREQUENCY**) instructs the program to print also the maximum mode shape value along each node circle for every requested mode, otherwise, only the actual mode shapes along nodal circles at requested circumferential angles will be printed.

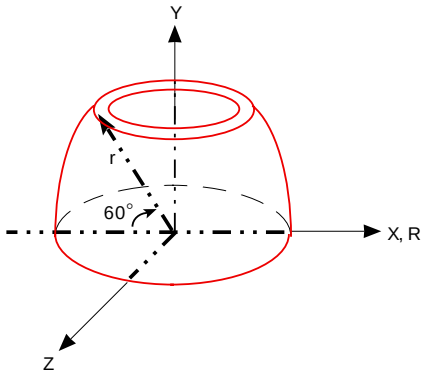
The following example, together with verification problems F25 and F26, illustrate the proper use of commands pertinent to the non-axisymmetric mode shape calculation.




Example 5: Frequency Analysis of a Circular Vessel

Evaluating the first five frequencies and mode shapes of the second harmonic (with geometric stiffening effect).

Figure 13-16.



Given:

E	= 30,000 ksi
r	= 10 in
Poisson Ratio	= 0.3

GEOSTAR Commands

The following commands cover the modeling of the problem.

Geo Panel: Propsets > **Element Group**
EGROUP,1,SHELLAX,0,0,0,0,0,0,0,

(Defining the element group as SHELLAX)

Geo Panel: Propsets > **Material Property**
MPROP,1,EX,3.E7,NUXY,.3,DENS,0.1,

(Defining the material properties)

Geo Panel: Propsets > **Real Constant**
RCONST,1,1,1,1,0.1,

(Defining the thickness of SHELLAX)

Geo Panel: Geometry > GRID > **Plane**
PLANE,Z,0,1,

Use View icon to set X-Y view
VIEW,0,0,1,0,

Geo Panel: Geometry > POINTS > **Define**
PT,1,0,0,0,
PT,2,10,0,0,

Geo Panel: Geometry > CURVES > CIRCLES > **Circle in Plane**
CRPCIRCLE,1,1,2,10,60.,1,

(Defining the circular geometry of SHELLAX)

Geo Panel: Meshing > PARAMETRIC MESH > **Curves**
M_CR,1,1,1,2,30,1,

(Generating 30 elements)

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > **Define Nodes**
DND,1,ALL,0,1,1,
DND,2,RY,0,31,1,RX,,

(Apply the displacement constraints)

Use Auto Scaling icon
SCALE,0,

Geo Panel: Analysis > OUTPUT OPTIONS > **Set Print Options**
PRINT_OPS,1,0,0,1,0,1,0,0,1,0,

(Requesting maximum response at each harmonic)

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Frequency Options**
A_FREQ,5,S,16,,,,1,,,,,,,,

(Specifying 5 frequencies and applying Subspace method with geometric stiffening effect)

Geo Panel: LoadsBC > STRUCTURAL > FORCES > **Define Nodes)**
FND,31,FY,100,31,1

(Define an axisymmetric load in axial direction to account for geometric stiffening)

Commands Pertinent to Asymmetric Mode Shapes

Geo Panel: Analysis > STATIC > **Asymmetric Load Options**
Asymmetric loading option > **Yes**

(Activating the asymmetric type)

Harmonic type > **GEN**

(Irrelevant; use default)

Beginning Harmonic number > **3**

(Consider the second harmonic)




Ending Harmonic number > **3**

(Irrelevant; use default)

Starting angle for output > **0**

(Start at 0 circumferential angle for output results)

Ending angle for output > **360**

(End at angle 360°)

Increment angle for output >**10**

Geo Panel: Analysis > FREQUENCY/BUCKLING > **Run Frequency**

(Perform frequency analysis)

Geo Panel: Analysis > STATIC > **Run Stress Analysis**

(To evaluate stresses over the mode shapes)

Maximum stresses (along the circumferential direction) for each individual mode shape are evaluated and printed. These stress quantities represent the harmonic coefficients of each mode which, in reality, correspond to the maximum value of the harmonic terms along each circumferential (node) circle.

3D Plots

The resolution in the circumferential direction is controlled by the angle increment in the **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command. Use the following sequence of commands to plot the first mode shape:

Geo Panel: Meshing > ELEMENTS > **Set Element Plot**
SETEPLOT, , , 1;

Click on View icon to set isometric view
VIEW, 1, 1, 1, ,

Geo Panel: Results > PLOT > **Deformed Shape**
DEFPLOT, 1;

Click on Perfect Scale icon
PSCALE;

The third mode may be plotted as follows.

Geo Panel: Results > PLOT > **Deformed Shape**
DEFPLOT, 3;

Figure 13-17 shows that the first mode along the wall of the vessel (meridian direction) is somewhat similar to that of a cantilever beam. In the circumferential

direction however the vessel deformed in three distinct modes demonstrating that the third harmonic is present. View from the Y-direction (**VIEW, 0, 1, 0;**) to get a better view.

Figure 13-17. First Mode of Third Harmonic

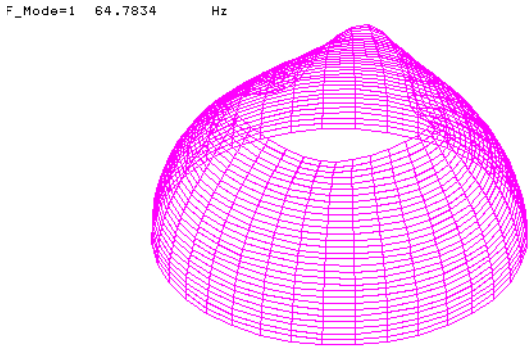
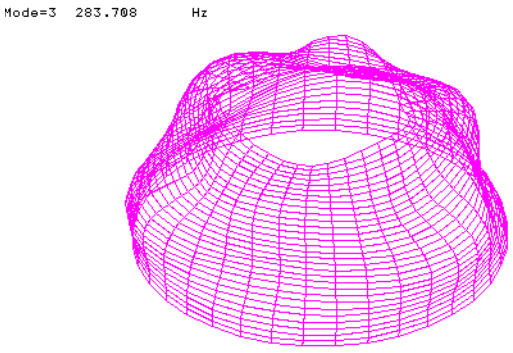


Figure 13-18 shows that the wall of the vessel deforms (in the meridian direction) similar to the third mode of a cantilever beam while the three distinct oscillations are maintained in the circumferential direction.

Figure 13-18. Third Mode of Third Harmonic



Non-Axisymmetric Buckling Analysis

Buckling analysis of axisymmetric models is limited to cases involving axisymmetric loads. The program marches through the calculation of buckling loads (and modes) incrementing the harmonic number from the specified minimum number until the one for which the minimum load value occurs is determined. The buckling load factor and the corresponding mode shape (at requested circumferential angles) as well as the maximum mode shape at each circumferential node circle are written in the output file.

In the expanded form (requested by the "detailed print option" flag in the **PRINT_OPS** (Analysis > OUTPUT OPTIONS > **Set Print Options**) command), the program considers all harmonics in the requested range and writes all the results in the output file.

The program calculates the first numerical eigenvalue of each harmonic which corresponds to the lowest buckling load. The actual buckling load is found by multiplying the calculated buckling load factor by the applied load.

The commands essential to set up the asymmetric buckling computation are described below. The model is assumed to have been defined.

1. **EGROUP** Command:

   Similar to the static analysis, the Asymmetric Option must be assigned for PLANE2D element (for the SHELLAX element, no additional options are required to activate).

2. ANALYSIS Command:

   The **A_ASYM** (Analysis > STATIC > **Asymmetric Load Options**) command has a similar function as in the case of static analysis, except that:

   a. Choice of harmonic number is irrelevant here. Use any option.

   b. The beginning and ending harmonics specify the range of harmonics for buckling analysis.

   The prompts for angles are applicable to the results written in the output files for the buckling mode shapes.




3. LISTING Commands:

   Use the **"A_LIST, BUCKL"** (Analysis > **List Analysis Option**) command to list general assignments for buckling analysis, including assignments made by the **A_ASYM** command. Use the **FLIST** or **PLIST** (LoadsBC > STRUCTURAL > FORCE, PRESSURE > **List**) commands to list the applied buckling loads.

4. **PRINT_OPS** Command:

   Activating the "detailed print option" flag of this command instructs the program to calculate the buckling loads (and corresponding mode shapes) for all the harmonics specified in the range and print all the results in the output file, otherwise, the program stops when the minimum buckling load is found and the results will be available only for the corresponding harmonic in the output file. Contrary to the static analysis, there is no need for using the **ACTSET** (Control > ACTIVATE > **Set Entity**) command with options FCOS, FSIN and FCOEF, nor is there any need for the **CURDEF** (LoadsBC > FUNCTION CURVE > **Time/Temp Curve**) command.

Verification problems B14 and B15 illustrate the proper use of commands pertinent to the non-axisymmetric buckling mode calculation.

3D Plots

Buckling mode shapes in 3D may be plotted following the same procedure described for frequency analysis. The only difference is that all plots correspond to the first frequency in the axial direction (meridian direction) and the mode shape number prompt (in the **DEFPLOT** (Results > PLOT > **Deformed Shape**) command) corresponds to the harmonic number (frequency contents in the circumferential direction).







14 Large File Managements (for STAR and DSTAR)

Introduction

In running large models, there is often the problem of shortage of disk space. To overcome this problem, you may direct the program to create and store some of the files or even portions of a large file into other drives (directories) where there are available disk spaces. Dispersion of files in multiple drives is available primarily for such files as Stiffness, Consistent Mass, and Geometric Stiffness which could have unproportionally larger size than other files of the database. This option is available for the STAR and DSTAR modules which generate these files. However, it is also utilized by ASTAR when reading these files. Basically there are three options available to manage the database files as follows. Use the **SELDIR** (Control > MISCELLANEOUS > **Select Dir for Partition**) command to activate these options.

Option 1

This option is used when the general Stiffness, the Geometric Stiffness, and/or the Consistent mass files are so large that there is not enough disk space in the working directory to store them. If there are other drives available, then each one of the above files could be broken to several pieces and each piece could be written to a different drive. The size of each piece is automatically computed and is proportional to the available disk space in each drive. In addition to the local drives, you may take advantage of networking drives as well as removable drives.

Notes:

1. The Stiffness file and other similar files are structured based on memory blocks. A block is the smallest unit in these files and the default size of a block is equal to the available RAM (memory). Therefore, every individual file segment can contain only an exact number of blocks (with no fraction). For machines with large RAM (memory), the size of these blocks could be very huge, and therefore make it difficult to utilize the available disk spaces to accumulate the segmented files conveniently (i.e., when the available disk space of a participating drive is smaller than the default size of a block). Therefore, in using the **SELDIR** (Control, Miscellaneous, **Select Dir for Partition**) command you have the option of reducing the block size in order to make the files more manageable.
2. In addition to the mentioned three files (Stiffness, Geometric Stiffness, and Consistent mass) there are other temporary files which could be generated during the run of the program as a duplicate file in special cases. All these files have exactly the same size and will be segmented and generated in the requested directories, except for files: ORI.STF, ORI.MAS, and ORI.GST which are for the external use by the user (upon request) and are generated each in one piece in the current directory. For more description on these files and how to generate them in other drives see option 3.
3. The name of Stiffness matrix file for each segment consists of the PROBLEM NAME plus an extension ".Sij", where "ij" is a two digit number corresponding to the segmentation number (for example, the third segment has an extension in the form of ".S03"). Similarly, the Consistent mass as well as the Geometric stiffness file are segmented to files with extension ".Gij".

Option 2

This option is similar to option 1, except that the user is able to assign the size of each segment.

Notes:

1. In Option 1, all the segment size allocations are made automatically based on the available disk spaces. However, for this option the user should be aware (for the accurate estimation of the required drive disk space) of the temporary files which will be created by the program in certain situations and will be also segmented in the same way. These temporary files, which are deleted immediately after the problem is solved, are:

a. Files with Problem name and extension ".Kij". The above files (with "ij" corresponding to the segment number) are duplicate of geometric stiffness files and are generated when one of the following conditions exists in a multi-block buckling problem:

i. There is a frequency shift.

ii. There are nonzero prescribed displacements.

b. Files with problem name and extension ".Uij". These files are generated in the Buckling problems as duplicate of the Geometric stiffness and in the frequency problems with Consistent mass option as the duplicate of Mass matrix when the Sturm Sequence Check is requested. (Also when in frequency problems, Modal Acceleration Method, MAM, flag is on together with the Shift flag.)

Option 3

In this option, the user may request generation of certain files in other directories. These files will not be segmented to smaller files and the entire files will be created at the assigned directories. These files are:

1. Stiffness Matrix file (Extension: .STF)
2. Geometric Stiffness Matrix file (Extension: .ST2)
3. Consistent Mass Matrix file (Extension: .ST2)

There are also similar files for external use which are generated upon request (flag in the **PRINT_OPT** command) and can be transferred to specified directories with the following complete names:

4. ORI.STF (Stiffness Matrix)
5. ORI.MAS (Consistent Mass Matrix)
6. ORI.GST (Geometric Stiffness Matrix)

In addition to the above files there are some temporary files which have the same size as the above files and are deleted upon the termination of the program run. These files can also be transferred to specified directories and their complete names are:

7. KLIN.STF

The above file is a duplicate of Geometric Stiffness file and is generated under the same conditions explained earlier for files with extension ".Kij" in the Note

for Option two. If this file is to be generated, it will be generated in the same directory for which the actual Geometric Stiffness file will be generated.

8. TMP.STU

The above file is generated as a duplicate of Geometric Stiffness file and has the same function as the files with extension ".Uij" have in the previous options (as described earlier). For this file (contrary to the "KLIN.STF file) the user may select a specific directory path.

The above eight files have the same size and may be generated in the assigned directories (drives).




A Units

Introduction

A brief description for the two commonly used systems of units in structural analysis is presented in this appendix. These are: the international system of units (SI), and the U.S. customary units. In COSMOS/M, you can adopt any system of units, including standard and non-standard systems. However, units must be consistent and you should remember to interpret the output accordingly. Failure to use consistent units could result in incorrect and misleading results.

This appendix presents information about units for analysis using the Basic System *only*. For other types of analysis units, please refer to Appendix B of the COSMOS/M User Guide.

International System of Units

The basic units in this system are the units of length, mass and time and are, respectively, meter (m), kilogram (kg), and second (s). Other units are derived from these basic units. Principal SI units used in structural mechanics are illustrated in Table A-1.

Table A-1. Principal SI Units Used in Structural Mechanics

Quantity	Unit	Symbol
Length	Meter	m
Mass	Kilogram	kg
Time	Second	s
Area	Square meter	m^2
Solids volume	Cubic meter	m^3
Liquid volume	Liter	$L = 10^{-3} m^3$
Velocity	Meter per second	m/s
Acceleration	Meter per second squared	m/s^2
Angle	Radian	rad
Angular velocity	Radians per second	rad/s
Angular acceleration	Radians per second squared	rad/s^2
Density	Kilogram per cubic meter	kg/m^3
Force	Newton	$N = kg\ m/s^2$
Moment of a force	Newton-meter	N m
Stress and pressure	Pascal	$Pa = N/m^2$
Frequency	Hertz	Hz = cycles/s
Impulse	Newton-second	N s
Work	Joule	J = N m
Power	Watt	W = J/s
Thermal conductivity (K)	Watt per meter per degree centigrade	W/m °C
Specific heat (C)	Joule per kilogram per degree centigrade	J/kg °C
Convection film coefficient	Watt per meter squared per degree centigrade	W/m^2 °C
Heat generation per source (Q)	Watt	W
Heat flux (heat generation per area)	Watt per meter squared	W/m^2
Heat generation per unit volume (QE)	Watt per cubic meter	W/m^3

U.S. Customary System of Units

The basic units in this system are the foot (ft) for length, the pound (lb) for weight, and the second (s) for time. The weight (not mass) is a basic system, contrary to the SI system. The system is gravitational (and not absolute) since the weight of a body depends on the value of the acceleration of gravity. For instance, the weight of a body varies slightly on Earth and the same weight on Earth is about six times its weight on the moon. The pound is defined as the weight of a platinum standard, called the standard pound, at sea level and latitude of 45 degrees.

While the standard pound is used as a unit of mass in many commercial transactions, it should not be so used in engineering practice. The mass in the U.S. customary system is the slug. The slug is defined as the mass which attains a unit acceleration (ft/s^2) when subjected to a unit force (lb). Principal U.S. customary units used in mechanics and their equivalent in the SI system are shown in the table below:




Table A-2. Conversion Table for U.S. Customary and SI Units

Quantity	U.S. Customary Unit	SI Equivalent
Length	ft in. mi.	0.3048 m 25.40 mm 1.609 km
Area	ft^2 in^2	0.0929 m^2 645.2 mm^2
Volume	ft^3 in^3	0.02832 m^3 16.39 cm^3
Liquid Volume	gal qt	3.785 L 0.9464 L
Velocity	ft/s in/s mi/h (mph) mi/h	0.3048 m/s 0.0254 m/s 0.4470 m/s 1.609 km/h
Acceleration	ft/s^2 in/s^2	0.3048 m/s^2 0.0254 m/s^2
Mass	oz mass lb mass slug ton	28.35 g 0.4536 kg 14.59 kg 907.2 kg
Force	kip lb weight oz weight	4.448 kN 4.448 N 0.2780 N
Moment of a force	lb ft lb in	1.356 N m 0.1130 N m
Area moment of inertia	in^4	0.4162 x 10^6 mm^4
Mass moment of inertia	lb ft s^2	1.356 kg m^2
Stress and pressure	lb/ft^2 lb/in^2 (psi)	47.88 Pa 6.895 kPa
Work or energy	ft lb	1.356 J
Impulse or momentum	lb s	4.448 N s
Power	ft lb/s	1.356 W

Engineering Prefixes

Multiple and sub multiple units commonly used in engineering practice are shown in the table below:

Table A-3. Common Engineering Prefixes and Symbols

Multiplication	Prefix	Symbol
10^{12}	tetra	T
10^{9}	giga	G
10^{6}	mega	M
10^{3}	kilo	k
10^{2}	hekto	h
10^{1}	deca	da
10^{-1}	deci	d
10^{-2}	centi	c
10^{-3}	milli	m
10^{-6}	micro	m
10^{-9}	nano	n
10^{-12}	pico	p
10^{-15}	femto	f
10^{-18}	atto	a

Consistent Units

Table A-4 shows the proposed consistent units for analysis using the Basic System.

Table A-4. Consistent Units for Linear Static Analysis

Description	COSMOS Name	*FPS[1] (gravitational)	*SI[2] (absolute)	*MKS[3] (gravitational)	CGS[4] (absolute)
Measure					
Length	X, Y, Z	in	m	cm	cm
Material Properties					
Elastic Modulus	EX, EY, EZ	lbs/in^2	Newton/m^2 or Pascal	kg/cm^2	dyne/cm^2
Shear Modulus	GXY, GYZ, GXZ	lbs/in^2	N/m^2 or Pa	kg/cm^2	dyne/cm^2
Poisson's Ratio	NUXY, NUYZ, NUXZ	in/in (no units)	m/m (no units)	cm/cm (no units)	cm/cm
Mass Density	DENS	lbs sec^2/in^4	kg/m^3	kg sec^2/cm^4	g/cm^3
Coeff. of Thermal Expansion	ALPX, ALPY, ALPZ	in/(in °F)	m/(m °K)	cm/(cm °C)	cm/(cm °K)
Material Angle	Beta	degree	degree	degree	degree
Anisotropic Material Stiffness	MC11,…,MC66 (total of 21 constants)	lbs/in^2	Newton/m^2	kg/cm^2	dyne/cm^2
Anisotropic Material Compliance	MC11,…,MC66 (total of 21 constants)	in^2/lbs	m^2/Newton	cm^2/kg	cm^2/dyne
Piezoelectric Material Constants	PC11,…,PC63 (total of 18 constants)	----	Volt/m	----	Volt/cm
Dielectric Material Constants	DC11,…,PC33 (total of 6 constants)	----	Farad/m	----	10^{-13} Farad/cm
Loads and Boundary Conditions					
Temperature		°F	°K	°C	°K
Translational Displacements	UX, UY, UZ	in	m	cm	cm
Rotational Displacements	RX, RY, RZ	radians	radians	radians	radians
Forces (nodal)	FX, FY, FZ	lbs	Newton	kg	dyne
Moments (nodal)	MX, MY, MZ	in lbs	m N	cm kg	cm dyne
Pressure	P	lbs/in^2	N/m^2 or Pa	kg/cm^2	dyne/cm^2
Distributed Beam Load	PB	lbs/in	N/m	kg/cm	dyne/cm

Table A-4. Consistent Units for Linear Static Analysis (Concluded)

Description	COSMOS Name	*FPS[1] (gravitational)	*SI[2] (absolute)	*MKS[3] (gravitational)	CGS[4] (absolute)
Loads and Boundary Conditions (Continued)					
Linear Acceleration	ACEL	in/sec^2	m/sec^2	cm/sec^2	cm/sec^2
Angular Velocity	OMEGA, CGOMEGA	rad/sec	rad/sec	rad/sec	rad/sec
Angular Acceleration	DOMEGA, DCGOMEGA	rad/sec^2	rad/sec^2	rad/sec^2	rad/sec^2
Results					
Reaction Forces	UX, UY, UZ	lbs	Newton	kg	dyne
Reaction Moments	RX, RY, RZ	in lbs	m N	cm kg	cm dyne
Displacements	UX, UY, UZ, RES	in	m	cm	cm
Stresses	SX, SY, SZ, TXY, TYZ, TXZ, P1, P2, P3, VON, INT	lbs/in^2	N/m^2 or Pa	kg/cm^2	dyne/cm^2
Strains	EPSX, EPSY, EPSZ, GMXY, GMYZ, GMXZ, ESTRN	in/in (no units)	m/m (no units)	cm/cm (no units)	cm/cm (no units)

* Units are consistent with the COSMOS/M material database.

1 FPS refers to the U.S. customary system of units.

2 SI refers to the International system of units.

3 MKS refers to the Metric system of units.

4 CGS refers to the French system of units.




B Summary of Commands

Introduction

This appendix presents a brief description of all commands applicable for performing linear static, buckling, and modal analysis. For a complete description of commands, please refer to the COSMOS/M Command Reference Manual.

The format of the descriptions in the following pages is as shown below:

Cryptic Command Name	Command Path	Brief Command Description
ACEL	LoadsBC > STRUCTURAL > GRAVITY > Define Acceleration	Define acceleration.
ACTDIS	Results > PLOT > Displacement	Load the specified displacement component into the plot buffer.

As seen above, all commands are shown in bold characters, and the corresponding menu where these commands can be found (referred to as the command path) are separated by horizontal arrows.




Cryptic Command Name	Command Path	Brief Command Description
ACEL	LoadsBC > STRUCTURAL > GRAVITY > Define Acceleration	Define acceleration.
ACTDIS	Results > PLOT > Displacement	Load the specified displacement component into the plot buffer.
ACTPOST	Results > SET UP > Set Postprocess Type	Specify the type of analysis for postprocessing.
ACTSTN	Results > PLOT > Strain	Load the specified strain component into the plot buffer.
ACTSTR	Results > PLOT > Stress	Load the specified stress component into the plot buffer.
ACTUSRPLOT	Display > XY PLOTS > Activate User Plot	Activates a user plot for postprocessing.
ADAPTIVE	Analysis > STATIC > Adaptive Method	Select (H, P, or HP) methods for static analysis.
ANIMATE	Results > PLOT > Animate	Animate the deformations and mode shapes of structures.
ASMESECDEF	Analysis > STATIC > ASME CODE > Define ASME Section	Specify a section for stress evaluation according to ASME code.
ASMESECLIST	Analysis > STATIC > ASME CODE > List ASME Section	List the specified ASME sections.
ASMESECDEL	Analysis > STATIC > ASME CODE > Delete ASME Section	Delete the specified ASME sections.
AVERAGE	Results > Average Nodal Stress	Specify the procedure to calculate average nodal stresses.
A_BUCKLING	Analysis > FREQUENCY/BUCKLING > Buckling Options	Specify details for the buckling analysis.
A_FREQUENCY	Analysis > FREQUENCY/BUCKLING > Frequency Options	Specify details for the frequency analysis.
A_LIST	Analysis > List Analysis Option	List active options for various types of analyses.
A_STATIC	Analysis > STATIC > Static Analysis Options	Specify details of analysis for the static module.
A_STRESS	Analysis > STATIC > Stress Analysis Options	Specify details of analysis for the stress module.




Cryptic Command Name	Command Path	Brief Command Description
BEAMRESLIS	Results > LIST > Beam End Force	List the nodal forces, moments and stresses for beam elements.
BEAMRESMAX	Results > EXTREMES > Beam End Force	List the extreme nodal forces and stresses for beam elements.
BMSECDEF	Propsets > Beam Section	Define standard beam cross section dimensions and real constants.
BMSECLIST	Edit > LIST > Beam Sections	Delete standard beam sections and the associated real constants.
BONDDEF	LoadsBC > STRUCTURAL > BONDING > Define Bond Parameter	Bond various types of structural elements.
CGLOC	LoadsBC > STRUCTURAL > GRAVITY > Orig of Inertial Sys	Define the origin of the inertial reference frame.
CGOMEGA	LoadsBC > STRUCTURAL > GRAVITY > Avg. Vel. of GL CS	Define angular velocity of the global Cartesian system.
CPCNS	LoadsBC > STRUCTURAL > COUPLING > Define Constraint Set	Define a point-to-(point, curve, or surface) constraint.
CPCNSDEL	LoadsBC > STRUCTURAL > COUPLING > Delete Constraint Set	Delete point-to-(point, curve, or surface) constraints.
CPCNSLIST	LoadsBC > STRUCTURAL > COUPLING > List Constraint Set	List point-to-(point, curve, or surface) constraints defined using the CPCNS command.
CPDOF	LoadsBC > STRUCTURAL > COUPLING > Define DOF Set	Define a pattern of coupled degrees of freedom.
CPDOFDEL	LoadsBC > STRUCTURAL > COUPLING > Delete DOF Set	Delete sets of coupled degrees of freedom.
CPDOFLIST	LoadsBC > STRUCTURAL > COUPLING > List DOF Set	List coupled degrees of freedom.
CPDOFPLOT	LoadsBC > STRUCTURAL > COUPLING > Plot DOF Set	Plot defined coupled degrees of freedom.
CPEQN	LoadsBC > STRUCTURAL > COUPLING > Define Constraint Eq.	Define a constraint equation.
CPEQNDEL	LoadsBC > STRUCTURAL > COUPLING > Delete	Delete coupling equations.
CPEQNLIST	LoadsBC > STRUCTURAL > COUPLING > List Constraint Eq.	List coupling equations defined using the CPEQN command.




Cryptic Command Name	Command Path	Brief Command Description
CPEQNVAL	LoadsBC > STRUCTURAL > COUPLING > Constr Eq. Constant	Define the right hand side of a constraint equation.
CRACK	Analysis > STATIC > CRACK > Define Crack	Define a crack set and specifies the corresponding nodes.
CRACKDEL	Analysis > STATIC > CRACK > Delete Crack	Delete a pattern of crack sets.
CRACKLIST	Analysis > STATIC > CRACK > List Crack	List a pattern of crack sets and the associated nodes.
CURDEF	LoadsBC > FUNCTION CURVE > Time/Temp Curve	Define (time, temperature, or B-H) curves.
CURDEL	LoadsBC > FUNCTION CURVE > Delete Time/Temp Curve	Delete a pattern of previously defined curves.
CURLIST	LoadsBC > FUNCTION CURVE > List Time/Temp	List previously defined curves.
DATA_CHECK	Analysis > Data Check	Check element groups, material and real constant sets.
DCDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Curves	Delete displacements at all nodes associated with a curve(s).
DCGOMEGA	LoadsBC > STRUCTURAL > GRAVITY > Ang. Acc. of GL CS	Define angular acceleration of the global Cartesian system.
DCR	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Curves	Specify displacements at nodes associated with curves.
DCT	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Contours	Specify displacements at nodes associated with contours.
DCTDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Contours	Delete displacements at all nodes associated with a contour(s).
DEFPLOT	Results > PLOT > Deformed Shape	Load and plot the deformed configuration.
DISLIST	Results > LIST > Displacement	Displacement components.
DISMAX	Results > EXTREMES > Min/Max Displacement	List the extreme values of displacements.
DISPLOT	Results > PLOT > Displacement	Plot the displacement component loaded into plot buffer.
DLIST	LoadsBC > STRUCTURAL > DISPLACEMENT > List	List prescribed nodal displacements on the screen.

Cryptic Command Name	Command Path	Brief Command Description
DND	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Nodes	Specify displacements at a pattern of nodes.
DNDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Nodes	Delete prescribed displacements at a pattern of nodes.
DOMEGA	LoadsBC > STRUCTURAL > GRAVITY > Angular Acceleration	Define angular acceleration.
DPDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Points	Delete displacements at nodes associated with keypoints.
DPLOT	LoadsBC > STRUCTURAL > DISPLACEMENT > Plot	Plot prescribed nodal displacements on the screen.
DPT	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Points	Specify displacements at nodes associated with keypoints.
DRDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Regions	Delete displacements at all nodes associated with a region(s).
DRG	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Regions	Specify displacements at nodes associated with regions.
DSDEL	LoadsBC > STRUCTURAL > DISPLACEMENT > Delete Surfaces	Delete displacements at all nodes associated with a surface(s).
DSF	LoadsBC > STRUCTURAL > DISPLACEMENT > Define Surfaces	Specify displacements at nodes associated with surfaces.
EGDEL	Edit > DELETE > Element Groups	Delete element groups from the database.
EGLIST	Edit > LIST > Element Groups	List defined element groups on the screen.
EGROUP	Propsets > Element Group	Define an element group.
EMFREAD	LoadsBC > LOAD OPTIONS > Read EMag Force	Enable magneto-structural coupling.
EM_MODEL	Analysis > ELECTRO MAGNETIC > Conductor Model	Specify stationary or uniformly moving conductors.
EM_OUTPUT	Analysis > ELECTRO MAGNETIC > Output Options	Control the printing intervals of results.
EPROPCHANGE	Propsets > Change El-Prop	Change the property set association for elements.
EPROPSET	Propsets > New Property Set	Assign attributes to elements generated from existing ones.




Cryptic Command Name	Command Path	Brief Command Description
FCDEL	LoadsBC > STRUCTURAL > FORCE > Delete Curves	Delete forces at all nodes associated with a curve(s).
FCR	LoadsBC > STRUCTURAL > FORCE > Define Curves	Specify forces at all nodes associated with a curve(s).
FCT	LoadsBC > STRUCTURAL > FORCE > Define Contours	Specify forces at all nodes associated with a contour(s).
FCTDEL	LoadsBC > STRUCTURAL > FORCE > Delete Contours	Delete forces at all nodes associated with a contour(s).
FLIST	LoadsBC > STRUCTURAL > FORCE > List	List prescribed nodal forces on the screen.
FLOWLIST	Results > LIST > Flow Result	List the quantities related to a fluid flow analysis.
FLOWMAX	Results > EXTREMES > Min/Max Flow	List the extreme values of flow components.
FLOWPLOT	Results > PLOT > Fluid Flow	Plot the flow component loaded into the plot buffer.
FLOWPROP	Results > LIST > Flow Properties	List film coefficient and mass flow rate.
FND	LoadsBC > STRUCTURAL > FORCE > Define Nodes	Specify forces at a pattern of nodes.
FNDEL	LoadsBC > STRUCTURAL > FORCE > Delete Nodes	Delete forces at a pattern of nodes.
FPDEL	LoadsBC > STRUCTURAL > FORCE > Delete Points	Delete forces at nodes associated with keypoints.
FPLOT	LoadsBC > STRUCTURAL > FORCE > Plot	Plot prescribed nodal forces on the screen.
FPT	LoadsBC > STRUCTURAL > FORCE > Define Points	Specify forces at nodes associated with keypoints.
FRDEL	LoadsBC > STRUCTURAL > FORCE > Delete Regions	Delete forces at all nodes associated with a region(s).
FREQLIST	Results > LIST > Natural Frequency	List the natural frequencies of the model.
FRG	LoadsBC > STRUCTURAL > FORCE > Define Regions	Specify forces at all nodes associated with a region(s).
FSDEL	LoadsBC > STRUCTURAL > FORCE > Delete Surfaces	Delete forces at all nodes associated with a surface(s).




Cryptic Command Name	Command Path	Brief Command Description
FSF	LoadsBC > STRUCTURAL > FORCE > Define Surfaces	Specify forces at all nodes associated with a surface(s).
GRVLIST	LoadsBC > STRUCTURAL > GRAVITY > List Gravity Load	List components of gravity and centrifugal loading.
IDRESULT	Results > PLOT > Identify Result	Display the location and the value of the plotted quantity.
INITXYPLOT	Display > XY PLOTS > Initialize	Restore the default settings for x-y plots.
ISOPLOT	Results > PLOT	Plots isoplanes for postprocessing quantity.
J_INTCRDEF	Analysis > STATIC > J_Integral Curve > Define Curve	Define J_ Integral paths.
J_INTCRDEL	Analysis > STATIC > J_Integral Curve > Delete Curve	Delete J_ Integral paths.
J_INTCRLIST	Analysis > STATIC > J_Integral Curve > List Curve	List J_ Integral paths.
J_INTCRPLOT	Analysis > STATIC > J_Integral Curve > Plot Curve	Plot J_ Integral paths.
LCCOMB	Results > Combine Load Case	Combine the response of desired load cases.
LCLIST	Analysis > STATIC > List Load Case	List load cases.
LCSET	Analysis > STATIC > Activate Load Case	Activate load cases for static analysis.
LSECPLOT	Results > PLOT > Path Graph	Plot path variation graphs of post-processing quantity.
MAKE_CYCLIC	LoadsBC > FUNCTION CURVE > Repeat Time/Temp	Repeat a pattern of (time, temperature, or B-H) curves.
MASSPROP	Control > MEASURE > Find Mass Property	List mass property information for a pattern of elements.
MPDEL	Edit > DELETE > Material Props	Delete material property sets from the database.
MPLIST	Edit > LIST > Material Props	List defined material property sets on the screen.
MPROP	Propsets > Material Property	Define a material property set.
NTCDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Curves	Delete temperatures at all nodes associated with a curve(s).




Cryptic Command Name	Command Path	Brief Command Description
NTCR	LoadsBC > THERMAL > TEMPERATURE > Define Curves	Specify a temperature at all nodes associated with a curve(s).
NTCT	LoadsBC > THERMAL > TEMPERATURE > Define Contours	Specify a temperature at all nodes associated with a contour(s).
NTCTDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Contours	Delete temperatures at all nodes associated with a contour(s).
NTLIST	LoadsBC > THERMAL > TEMPERATURE > List	List prescribed nodal temperatures.
NTND	LoadsBC > THERMAL > TEMPERATURE > Define Nodes	Specify a temperature at a pattern of nodes.
NTNDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Nodes	Delete prescribed temperatures at a pattern of nodes.
NTPDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Points	Delete temperatures at nodes associated with keypoints.
NTPLOT	LoadsBC > THERMAL > TEMPERATURE > Plot	Plot a symbol at nodes with specified temperatures.
NTPT	LoadsBC > THERMAL > TEMPERATURE > Define Points	Specify a temperature at nodes associated with keypoints.
NTRDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Regions	Delete temperatures at all nodes associated with a region(s).
NTRG	LoadsBC > THERMAL > TEMPERATURE > Define Regions	Specify a temperature at all nodes associated with a region(s).
NTSDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Surfaces	Delete temperatures at all nodes associated with a surface(s).
NTSF	LoadsBC > THERMAL > TEMPERATURE > Define Surfaces	Specify a temperature at all nodes associated with a surface(s).
NTVDEL	LoadsBC > THERMAL > TEMPERATURE > Delete Volumes	Delete temperatures at all nodes associated with a volume(s).
NTVL	LoadsBC > THERMAL > TEMPERATURE > Define Volumes	Specify a temperature at all nodes associated with a volume(s).
OMEGA	LoadsBC > STRUCTURAL > GRAVITY > Angular Velocity	Define angular velocity.
PBEDEL	LoadsBC > STRUCTURAL > PRESSURE > Delete Beam Elem Load	Delete beam element loading.

Cryptic Command Name	Command Path	Brief Command Description
PBEL	LoadsBC > STRUCTURAL > PRESSURE > Define Beam Elem Load	Specify beam element loading.
PBELIST	LoadsBC > STRUCTURAL > PRESSURE > List Beam Elem Load	List beam element loading.
PCDEL	LoadsBC > STRUCTURAL > PRESSURE > Delete Curves	Delete pressure on all elements associated with a curve(s).
PCR	LoadsBC > STRUCTURAL > PRESSURE > Define Curves	Specify pressure on all elements associated with a curve(s).
PEDEL	LoadsBC > STRUCTURAL > PRESSURE > Delete Elements	Delete pressure on a pattern of elements.
PEL	LoadsBC > STRUCTURAL > PRESSURE > Define Elements	Specify pressure on elements.
PICK_MAT	Propsets > Pick Material Lib	Pick a material from the COSMOS/M material library.
PICK_SEC	Propsets > AISC Sect. Table	Pick a standard section from the AISC tables.
PLIST	LoadsBC > STRUCTURAL > PRESSURE > List	List prescribed pressures on the screen.
PPLOT	LoadsBC > STRUCTURAL > PRESSURE > Plot	Plot prescribed pressures on the screen.
PRDEL	LoadsBC > STRUCTURAL > PRESSURE > Delete Regions	Delete pressure on all elements associated with a region(s).
PRG	LoadsBC > STRUCTURAL > PRESSURE > Define Regions	Specify pressure on all elements associated with a region(s).
PRINT_ELSET	Analysis > OUTPUT OPTIONS > Set Element Range	Define groups of elements for stress evaluation.
PRINT_NDSET	Analysis > OUTPUT OPTIONS > Set Nodal Range	Define groups of nodes for results evaluation.
PRINT_OPS	Analysis > OUTPUT OPTIONS > Set Print Options	Control writing of results to the output file.
PSDEL	LoadsBC > STRUCTURAL > PRESSURE > Delete Surfaces	Delete pressure on all elements associated with a surface(s).
PSF	LoadsBC > STRUCTURAL > PRESSURE > Define Surfaces	Specify pressure on all elements associated with a surface(s).




Cryptic Command Name	Command Path	Brief Command Description
RCDEL	Edit > DELETE > Real Constants	Delete real constant sets from the database.
RCLIST	Edit > LIST > Real Constants	List defined real constant sets on the screen.
RCONST	Propsets > Real Constant	Define a real constant set.
REACTION	Analysis > Reaction	Request reaction forces computation at all nodes.
RESULTS?	Results > Available Result	Results for postprocessing.
R_BUCKLING	Analysis > FREQUENCY/BUCKLING > Run Buckling	Perform buckling analysis.
R_CHECK	Analysis > Run Check	Check element's connectivity, aspect ratio, and other element attributes.
R_FREQUENCY	Analysis > FREQUENCY/BUCKLING > Run Frequency	Calculate natural frequencies and mode shapes.
R_STATIC	Analysis > STATIC > Run Static Analysis	Run linear static analysis.
R_STRESS	Analysis > STATIC > Run Stress Analysis	Perform stress analysis for linear static problems.
SECPLOT	Results > PLOT	Plot sections of postprocessing quantity.
SELDIR	Control > MISCELLANEOUS > Select Dir for Partition	Define allocation of disk space for large problem.
SELDIRLIST	Control > MISCELLANEOUS > List Dir for Partition	List disk allocation space for large problem.
SETERASE	Results > SET UP > Set Clear Screen Options	Provide the option to clear screen before new plots.
SETPLOT	Results > SET UP > Color/Value Range	Set the plot type and specify extreme values.
SETXYPLOT	Display > XY PLOTS > Set Plot Parameter	Set flags and parameters for x-y plots.
SMLIST	Results > LIST > Shear/Moment Value	List the shearing forces and bending moments for beam elements.
SMMAX	Results > EXTREMES > Shear/ Moment	List the extreme forces (over full length) for beam elements.




Cryptic Command Name	Command Path	Brief Command Description
SMPLOT	Results > PLOT > Shear Diagram	Plot shear and moment diagrams for beam elements.
SPDEL	Analysis > STATIC > SUBSTRUCTURE > Delete Super Element	Delete super elements from the database.
SPELE	Analysis > STATIC > SUBSTRUCTURE > Define Super Element	Define connectivity for a super element.
SPLIST	Analysis > STATIC > SUBSTRUCTURE > List Super Element	List super elements and their super nodes.
SPROT	Analysis > STATIC > SUBSTRUCTURE > Rotate Super Element	Rotate a super element in the global Cartesian system.
SPSTR	Analysis > STATIC > SUBSTRUCTURE > Set SubStr Analysis	Create super elements and defines type of analysis.
STNLIST	Results > LIST > Strain Component	List the strain components.
STNMAX	Results > EXTREMES > Min/Max Strain	List the extreme values of strains.
STNPLOT	Results > PLOT > Strain	Plot the strain component previously loaded into plot buffer.
STRAIN_OUT	Analysis > OUTPUT OPTIONS > Set Strain Output	Control writing of strain components in the output file.
STRESS	Analysis > STATIC > Activate Stress Calc	Activate stress calculations in the static module.
STRLIST	Results > LIST > Stress Component	List the stress components.
STRMAX	Results > EXTREMES > Min/Max Stress	List the extreme values of stresses.
STRPLOT	Results > PLOT > Stress	Plot the stress component previously loaded into plot buffer.
SUBMODEL	Analysis > STATIC > Define SubModel	Specify region(s) as submodel.
TEMPREAD	LoadsBC > LOAD OPTIONS > Read Temp as Load	Use information from a thermal to a linear static analysis.




Cryptic Command Name	Command Path	Brief Command Description
TREF	LoadsBC > LOAD OPTIONS > Reference Temp	Define the reference temperature for the model.
TUNIF	LoadsBC > LOAD OPTIONS > Uniform Temp	Define a uniform temperature for all nodes in the model.
USER_MAT	Propsets > User Material Lib	Pick a material from the user-created material library.
USRPLOT	Results > PLOT > User Result	Plots user-defined plot.
XYIDENTIFY	Display > XY PLOTS > Identify Point	Identify the coordinates of a point in the graph range.
XYLIST	Display > XY PLOTS > List Info	List the available information to produce x-y plots.
XYPLOT	Display > XY PLOTS > Plot Curves	Generate the time-history-like plots.
XYPTLIST	Display > XY PLOTS > List Points	List a pattern of points for loaded graphs.
XYRANGE	Display > XY PLOTS > Set Plot Range	Specify the graph ranges and scale factors.
XYREFLINE	Display > XY PLOTS > Set Reference Line	Plot a reference line parallel to the x or y axis.




C A Brief Theoretical Background for ASME Stress Code Evaluation

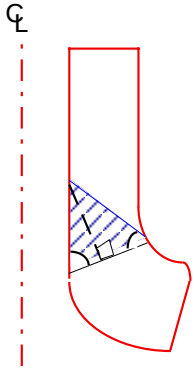
Introduction

This chapter contains additional details on ASME code, section III, for stress evaluation.

Section Orientation in Junctions

For sections in the nozzle to shell junctions, rational planes of bending should be approximated such that they are perpendicular to the mid-plane and have the same angle between the section and the surface on both sides. This is done by forming an isosceles triangle which has the section as its base and the mid-plane as its altitude as shown in the figure below. One side of the triangle is tangent to the fillet at the point of interest.

Figure C-1. Defining a Section in an Irregular Area



Stress Linearization

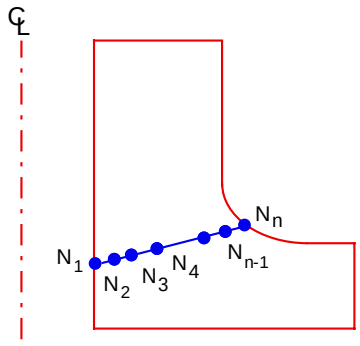
For plane stress, plane strain and axisymmetric structures, stress linearization is performed along a section path. A section is defined by nodes N_1 and N_n according to the following figure. The program interpolates n-2 equally spaced (integration) points along the path between nodes N_1 and N_n. For each point, structural elements are searched to identify an element which contains that point. Once the element is identified, the stresses at that point are interpolated linearly from the element corner nodes. These stresses are denoted as actual stresses.

The equivalent linearized stresses are found along a section by evaluating the membrane and bending stresses according to one of the following two methods.

Cartesian Formulation (Approximation)

This formulation is applicable to plane stress and plane strain problems as well as axisymmetric problems where the radial dependency could be ignored. For axisymmetric structures the radial dependency is due to the fact that there is more material at a greater radius than at a smaller one. This option is implemented into the calculation if the input quantity "curvature_radius" in command **ASMESECDEF** (Analysis > STATIC > ASME CODE > **Define ASME Section**) is set to zero (see the Axisymmetric Formulation section for the definition of curvature_radius).

Figure C-2. A Section with "n" Equally Spaced Integration Points Along its Path



The membrane components of the stresses are evaluated by integrating along the section path according to:

$$\begin{Bmatrix} \sigma_X^m \\ \sigma_Y^m \\ \sigma_Z^m \\ \sigma_{XY}^m \end{Bmatrix} = \frac{1}{\ell} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \begin{Bmatrix} \sigma_X^a \\ \sigma_Y^a \\ \sigma_Z^a \\ \sigma_{XY}^a \end{Bmatrix} d\eta \qquad \text{(C-1)}$$

where, superscripts *m* and *a* correspond to membrane and actual stresses, respectively, *l* is the length of the section, x and y are the local coordinates along and perpendicular to the section path (see figure below) with an origin at the mid-wall, and X, Y and Z are the global Cartesian coordinates. Membrane stresses are considered to be constant along the section.

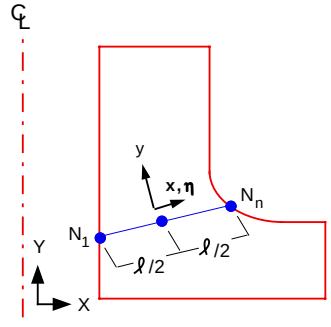
The bending component of stresses at position x along the section path is calculated according to:

$$\begin{Bmatrix} \sigma_X^b(X) \\ \sigma_Y^b(X) \\ \sigma_Z^b(X) \\ \sigma_{XY}^b(X) \end{Bmatrix} = \frac{12x}{\ell^3} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \begin{Bmatrix} \sigma_X^a \\ \sigma_Y^a \\ \sigma_Z^a \\ \sigma_{XY}^a \end{Bmatrix} \eta \, d\eta \qquad \text{(C-2)}$$

where superscript *b* corresponds to the bending stress.

The linearized stress at any point along the section is the sum of membrane and bending stresses.

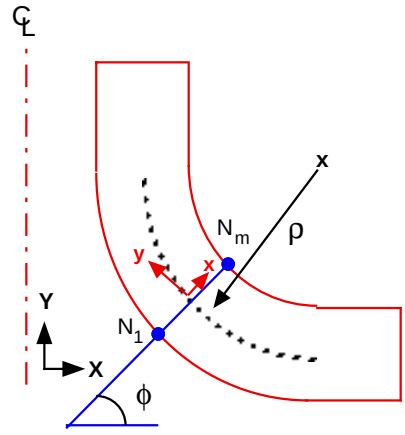
Figure C-3. Definition of the Section Local Coordinates



Axisymmetric Formulation

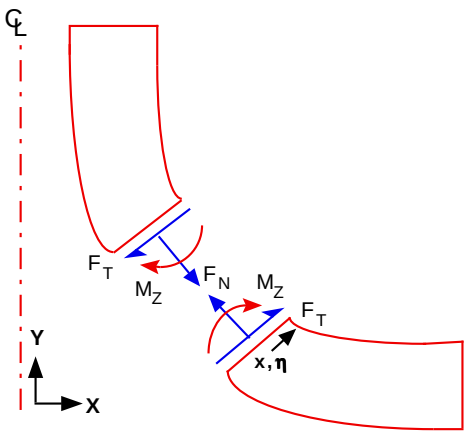
In this case the axisymmetric features of the model are more properly implemented in the formulation, for axisymmetric structures, than the Cartesian case. This option is considered in stress evaluation if a non-zero value is assigned to the input quantity "curvature_radius" in the command **ASMESECDEF** (Analysis > STATIC > ASME CODE > **Define ASME Section**). Curvature_radius corresponds to the radius of curvature of the average mid-wall centerline in the xy plane as represented by "ρ" in the figure below. A large value (or -1) for curvature_radius corresponds to straight walls (e.g., cylinder or cone).

Figure C-4. Curvature-Radius of the Mid-Wall



In order to find the linearized stresses, it is desired to obtain applied forces and moments along the section. The figure below represents a free-body diagram of the section. A right-handed local coordinate system, x, y, and z is established on the section with the origin at the mid-wall (same as in the two figures above). F_N and F_T correspond to the in-plane normal and shear forces on the section in y and x directions and M_Z is the bending moment.

Figure C-5. Applied Forces and Moments Along the Section



The three in-plane forces and moment on the section over a small sector ($\Delta\theta$) in the hoop direction are defined as:

$$\begin{Bmatrix} F_N \\ M_Z \\ F_T \end{Bmatrix} = \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \begin{Bmatrix} \sigma_y^a \\ (\eta - x_f)\,\sigma_y^a \\ \sigma_{xy}^a \end{Bmatrix} X\,\Delta\theta\,d\eta \qquad \text{(C-3)}$$

where, σy^a and σxy^a are the actual stresses along the section in local coordinates, X is the global coordinate (or equivalently the radius) of point being integrated along the section and x_f in local coordinates is the offset of the sector neutral axis from the center line.

From F_N in the above equation, the average normal membrane stress in the y-direction is computed once it is divided by the sector area (X_c is the global coordinate of the mid-section).

$$\sigma_y^m = \frac{1}{X_c\,\ell}\int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \sigma_y^a\,X\,d\eta \qquad \text{(C-4)}$$

The M_Z contribution to the normal bending stress in the y-direction is computed by applying the familiar relationship $s = M_z\,(x\text{-}x_f) / I$, with I as moment of inertia of the sector.

$$\sigma_y^b(x) = \frac{x - x_f}{X_c\left(\frac{\ell^2}{12} + x_f^2\right)\ell}\int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} (\eta - x_f)\,\sigma_y^a\,X\,d\eta \qquad \text{(C-5)}$$

An average membrane shear stress (xy component) is computed by dividing F_T (in Equation C-3) by the sector area, assuming that xy bending shear stress has insignificant contribution (since the shear stress distribution is assumed to be parabolic and equal to zero at the two free surface ends).

$$\sigma_{xy}^{m} = \frac{1}{X_c \ell} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \sigma_{xy}^{a} X \, d\eta \qquad \text{(C-6)}$$

Average membrane stress in the x-direction is computed by averaging the actual stresses along the section according to:

$$\sigma_{x}^{m} = \frac{1}{\ell} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \sigma_{x}^{a} \, d\eta \qquad \text{(C-7)}$$

The bending stress in the x direction (thickness direction) is ignored if in command **ASMESECDEF** (Analysis > STATIC > ASME CODE > **Define ASME Section**) the corresponding flag is set to one, otherwise it is approximated at the two ends as the difference of the actual and membrane stresses.

The hoop membrane and bending stresses are calculated by considering a small sector ($\Delta\phi$) in the XY plane. By integrating the total normal force on the sector and averaging it over the corresponding area, the average membrane stress (in hoop direction) is computed according to:

$$\sigma_{z}^{m} = \frac{1}{\ell} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} \frac{\sigma_{z}^{a}(\rho + \eta)}{\rho} \, d\eta \qquad \text{(C-8)}$$

For straight walls where ρ approaches infinity, Eq. (C-8) reduces to the familiar form.

The hoop bending stress is calculated by evaluating the applied bending moment on the sector. Once the bending moment is evaluated, the hoop bending stress is found analogous to that of the y direction bending stress.

$$\sigma_z^b(x) = \frac{x - x_h}{X_c\left(\frac{\ell^2}{12} + x_h^2\right)\ell} \int_{-\frac{\ell}{2}}^{\frac{\ell}{2}} (\eta - x_h)\frac{(\rho + \eta)}{\rho}\sigma_z^a \, d\eta \qquad \text{(C-9)}$$

where, $x_h = \frac{\ell^2}{12\rho}$

As mentioned before, the linearized stress at any point along the section is the sum of membrane and bending stresses.







D Troubleshooting

Introduction

This appendix lists, in alphabetical order, the error messages produced by the direct solvers for linear static analysis (STAR and STRESS) and frequency and buckling analyses (DSTAR). Messages from the two auxiliary models (PRE1 and RENUM) used in the basic system are also presented. Possible conditions leading to the message and the needed corrective measures are provided.

The RENUM Auxiliary Module

RENUM is called by STAR and DSTAR to renumber the nodes in order to minimize the bandwidth of the assembled stiffness matrix. Reduction in the bandwidth results in a substantial reduction in the computational time and resources required to solve the problem.

The PRE1.EXE Module

The PRE1 module is used by STAR and DSTAR to:

- Assign the proper index to each global degree of freedom (DOF).
- Compute the equivalent nodal forces for applied pressures for the static and buckling analyses as well as frequency analysis when the effect of loads (in-plane loading or stress stiffening effect) on the natural frequencies is considered.

Linear Static Analysis

Linear static analysis uses RENUM (optional), PRE1, STAR, and STRESS. The STAR.EXE module uses information prepared by PRE1.EXE and RENUM.EXE to:

- Compute the stiffness matrix of each element.
- Assemble the global stiffness matrix from the element stiffness matrices.
- Decompose the assembled matrix and solve for displacements at each degree of freedom for all active load cases.
- The STRESS module is then used to calculate stresses from displacement profiles.

When the in-plane stiffening effect is considered the above procedure will be different.

Frequency and Buckling Analyses

Frequency and buckling analyses use RENUM (optional), PRE1, STAR, and DSTAR. The steps are as follows:

- RENUM and PRE1 prepare the information as described above.
- STAR calculates the stiffness and mass matrices for each element and assembles and decomposes the global stiffness and mass matrices.
- DSTAR extracts the specified number of frequencies (frequency analysis), or critical load factors (buckling analysis) using the specified algorithm.
- STRESS (optional) is used to calculate stresses based on the normalized mode shapes. Note that the actual values are meaningless but the profile indicates the stress distribution when the structure is vibrating in a certain mode.

All The Nodes Defined For CE Constraints Should Be Either Inside Or Outside Of The Submodel For CE Constraint: <Number>

STAR.EXE Module

The nodes defining a Constraint Equation set (CE) should be either all inside or all outside of a submodel. Please check the mesh carefully. Define a new submodel and either include or exclude all those nodes from that submodel.

Define a new submodel which includes or excludes all these constraints.

All The Nodes Defined For PTL Constraints Should Be Either Inside Or Outside Of The Submodel For PTL Constraint: <Number>

STAR.EXE Module

The nodes defining a point-to-line constraint set should be either all inside or all outside of a submodel. The specified coupling set includes a node label which is not present in the submodel.

Define a new submodel which includes or excludes all these constraints

All The Nodes Defined For PTP Constraints Should Be Either Inside Or Outside Of The Submodel For PTP Constraint: <Number>

STAR.EXE Module

The nodes defining a point-to-point constraint set should be either all inside or all outside of a submodel.

Define a new submodel which includes or excludes all these constraints.

All The Nodes Defined For PTS Constraints Should Be Either Inside Or Outside Of The Submodel For PTS Constraint: <Number>

STAR.EXE Module

The nodes defining a point-to-surface constraint set should be either all inside or all outside of a submodel. Please check the mesh carefully.

Define a new submodel which includes or excludes all these constraints.

Allocated Memory Needs To Be Increased. Use RENUM Command To Increase The Memory By A Factor Of: <Factor>

RENUM.EXE Module

A shortage of the memory for this problem

Use the virtual memory by using the RENUM command (Analysis, Renumber) to apply the specified factor and rerun the problem. If the error message persists, then multiply with new factor, re-apply it in the RENUM command, and rerun the problem. See also the comments regrading the Error Message: "Not enough memory to run RENUM".

Angle Increment For Printout Is Zero

STAR.EXE, STRES.EXE or DSTAR.EXE Module

The requested angle increment, in circumferential direction, for evaluation of response in the analysis of your asymmetric problem is zero or very small (smaller than 1.0E-6)

Use the A_ASYM command (Analysis, Static, Asymmetric Load Options) to specify an acceptable value and try again.

Anisotropic Option Is Not Available For Fluid Elements

STAR.EXE, and STRES.EXE Module

This version of COSMOS/M does not support direct input of the elastic matrix for incompressible materials (fluid element option).

Change the element type from fluid to solid or use isotropic or orthotropic material properties.




Approximate Disk Space Required . . . (MEGABYTES) =<Size>. Available Free Disk Space . . . (MEGABYTES) = <Size>

STAR.EXE Module

The free disk space available in the working drive is smaller than the required size for this particular problem.

You may redirect storage to one or more other drives. Refer to the Large File Management chapter in the Basic System manual.

Approximate Total Size Of The Problem (Unit) = <Size> Available Free Disk Space (Unit) = <Size>

STAR.EXE Module

The available free disk space is not sufficient to run this problem.

Use the Large File Management utility to use one or more drives in storing the problem database. Refer to the Large File Management chapter in the Basic System manual.

At Least Two Sets Of Requested Plot Ranges Have Common Steps

STRES.EXE Module After Running Advanced Dynamic Analysis (DPOST.EXE).

This message indicates that at least two plot sets requested using the PD_PLOT command (Analysis, Post Dynamic, PD_OUTPUT, Set Plot Options) share the same time step.

Use the PD_PLTLIST command (Analysis, Post_Dynamic, PD_OUTPUT, List Plot Info) to examine requested plot sets and use the PD_PLOT command (Analysis, Post_Dynamic, PD_OUTPUT, Set Plot Options) to correct your input and then try the stress analysis again.

Bond Nodes Should Be Either Inside Or Outside Of The Submodel For Bond Constraint: <Number>

STAR.EXE Module

The nodes belonging to a Bond set should be either all inside or all outside of a submodel. Please check the mesh carefully.

Define a new submodel which includes or excludes all these constraints

Center Node Not Defined For a SHELL9 Element Used With 9-Node Option, at Element: <number> For Element Group: <group number>

STAR.EXE Module

The specified element belongs to a SHELL9 or SHELL9L group for which the 9-node element option has been selected. The ninth (center) node has not been defined.

Either re-define a center node for the element or change the corresponding element group to the "8-node option" using the EGROUP command (PropSets, Element Group).

Command ECHECK Needs To Be Issued To Correct The Wrong Collapsed Nodes Of Element Number <Number>

STAR.EXE or STRES.EXE Module

The connectivity of this element is invalid. For the SHELL9 elements, only the third, fourth and seventh nodes are allowed to collapse.

Use the ECHECK command (Meshing, Elements, Check Elements) to fix the connectivity of this element.

Composite Damping Not Available With Consistent Mass

STAR.EXE Module

You have requested frequencies using the consistent mass matrix formulation. DSTAR supports the composite damping option only with the lumped matrix formulation.

Use the A-FFEQUENCY command (Analysis, Frequency/Buckling, Frequency Options) to specify lumped mass formulation and try again.

Computation Of Displacement Not Performed/completed. Press Enter Key To Quit From The Program

STRES.EXE Module

It indicates that the pre-stress calculation (displacement calculation) has not been completed successfully.

If you are running the modules interactively (one by one outside GEOSTAR) make sure that you did not skip the PRE1.EXE and STAR.EXE modules. Check the output file to see whether displacement calculations were completed. The total elapsed time should be recorded in the output file for displacement calculations at the final stage of computation. You might have pressed the ESC key earlier to terminate the calculation of displacements. Rerun the problem from the beginning (displacements and stresses).

Convergence May Not Have Achieved For Harmonic <Number>

STAR.EXE Module

Convergence was not achieved for the specified harmonic number or you have two close modes and the program converged to the higher mode.

Use the A-FFEQUENCY command (Analysis, Frequency/Buckling, Frequency Options) to Increase the number of iteration, or introduce a small shift to avoid convergence to a higher harmonic.

Convergence Not Achieved For The Requested Tolerance Within Specified Number Of Iteration

DSTAR.EXE Module

Convergence has not been achieved with the specified tolerance and number of iterations.

Increase the number of iterations or relax the tolerance level and rerun the problem again.

Convergence Not Achieved For Requested Tolerance Within Specified Number Of Iteration. Save The Results (Y/N)?

DSTAR.EXE Module

For Subspace method, if the user's requested convergence is not achieved within the specified number of iteration, then by responding yes (pressing "Y" key) the results will be saved in data base, otherwise the program terminates without saving the results.

Displacement/Reaction Not Found From Old File For Node <Number>

STAR.EXE Module

This message occurs when trying to retrieve information for submodeling. The old file refers to the displacement file for the last run of the problem. The program gives this message when it can not retrieve results for the specified boundary node (at the interface between the submodel and the rest of the model).

Check your disk and your submodeling procedure and try again.

Displacement Not Available In Old File For Load Case <Number>

STAR.EXE Module

The displacement results are not available for the specified load case in the displacement file generated in the previous run (prior to running the current submodel).

You may have activated a new load case prior to running the current submodel for which no results are available from the previous run.

Do You Wish To Save The Current Iteration Results (Y/N)?

DSTAR.EXE Module

You may get this message if you press the Esc key during the execution of the DSTAR.EXE program.

If you press Y, the program will complete the current iteration and save it before quitting. The completion time depends on the size of the problem as well as the stage of calculation at which you have pressed the ESC key. If you press N, the computation will be terminated immediately. Note that even if you initially press Y, you can still terminate the program at any time by pressing the ESC key again.

8 Node Option Is Not Available Yet For SHELL9L(P)

STAR.EXE Module

You chose to use the P-method for the adaptive solution with this element. Currently, the 8-node SHELL9L element does not support the P- and HP-method. It only supports the H-method.

You may use the 9-node SHELL9L element or use the H-method.

Elastic Matrix Not Positive-Definite For Orth. Mat. Set <Number>

STAR.EXE Module

The orthotropic material properties of the specified set led to an elastic matrix that is not positive definite. The elastic matrix relates strain and stress tensors.

Check the validity of the material properties in the specified set and try again.

Element Area Is Zero, Element <Element>

STAR.EXE Module

The specified 2D element is invalid because its area is zero.

List the element and check the validity of its nodes and their connectivity.

Element Thickness Is Smaller Than Tolerance <Value>, Element <Number>

STAR.EXE or STRES.EXE Module

Element thickness is not defined or its defined value is smaller than the specified value.

List the element, check the associated real constant set. Modify the real constant set to include a valid value for thickness.

Element Volume Is Smaller Than Limit <Value> For Element <Number>

STAR.EXE or STRES.EXE Module

The volume of the specified element is smaller than the tolerance value. Elements smaller than the tolerance value cause numerical difficulty in the computation and should be avoided.

Refer to the message "Stop - Element area is smaller than tolerance <value>, element <number> for notes.

Error In Calculating The Direction Cosines For Pipe Element <Number>

STAR.EXE Module

For the specified PIPE element, an error was encountered during the computation of direction cosines relating local and global coordinate systems.

List the element and check its connectivity. In particular, check the validity of the third node relative to the two end nodes. The three nodes may not be collinear.

Error In Calculating Normal To The Side Of Element <Number>

STAR.EXE Module

The SOLIDL element may have inconsistent number of nodes on the top and bottom faces (i.e., one face may have collapsed nodes).

List the element and examine the validity of its nodes and their connectivity.

Error In Eigenvector Calculation. {PHIi} [M] {PHIi}=0

DSTAR.EXE Module

Error occurs during the normalization of eigenvectors with respect to mass. In the error message {PHIi} corresponds to the i-th. eigenvector and [M] to the mass matrix.

The multiplication expressed in the error message should always be a positive number.




Error - For Incompressible 4-node Fluid Element <Number> Shear Modulus (GXY) Must Be Smaller Than <Value>

STAR.EXE Module

For the specified 4-node Fluid plane element, the shear modulus is smaller than the limit value.

List the element and find out the associated material set. List the material set and modify the set to include a valid value for GXY.

ERROR - Less Number Of Nodes Than Required For Element Number <Number>

STAR.EXE Module

The number of nodes defining the specified element is insufficient.Check the mesh at the specified element.

List the specified element and make sure that it belongs to a proper element group. Redefine the element group to be valid for this element, or fix the element by redefining it or remeshing.

ERROR - Material Property Not Admissible For Element <Number>

STAR.EXE and STRES.EXE Module

The material set corresponding to the specified element has inconsistent material properties.

List the element and find out the associated material set. List the material set and modify it as needed to be admissible.

Error - Multi-Base Motion File (.MBS) Is Not Available

STRES.EXE Module

The *problem-name.MBS* file is generated during frequency analysis.

Make sure that the Multi-base motion input information is provided before running frequency analysis using the DSTAR.EXE module.

Error - No Master Nodes Defined For A Guyan Reduction

DSTAR.EXE Module

No master nodes defined.

Define the master nodes properly and rerun the problem.

ERROR - Non-existent Coupled Node <Number> For Set <Number>

STAR.EXE Module

The specified coupled set includes a node label which is not present in the model. The node may have been deleted after defining the set.

List the coupling set and modify it as needed using the Loads-BC, Structural, Coupling) menu.




ERROR - Not Enough Memory To Run Piezoelectric

STAR.EXE Module

In order to run this problem, you need to increase the amount of free memory.

Increase the size of the memory available by closing other applications. If the size of the RAM on your machine is small, you may increase the memory by installing extra RAM.

Error- Number Of Requested Steps For Plot Exceeds The Limit <Number>

STRES.EXE Module After Running Advanced Dynamic Analysis (DPOST.EXE).

The number of the requested steps for plot exceeded the specified limit.

Reduce the number of plots and rerun the STRESS.EXE module again.

Error In Opening .LC1 File As New

STAR.EXE Module

The file *problem-name.LC1* file is an internal file used to store displacement results of asymmetrical problems. The error message indicates that the file already exists.

Delete or rename the file and try again.

Error In Opening .LCD FIle As New

STAR.EXE Module

The file *problem-name.LCD* file is an internal file used to store displacement results. The error message indicates that the file already exists.

Delete or rename the file and try again.

Error In Opening The Plot File To Write Mass Participations

DSTAR.EXE Module

The problem-name.LCM is used to store the mass participation factors. An error was encountered in opening this file.

Check your disk space. The database may have been corrupted. Contact SRAC if the error persists.

Error In Opening The Stress File

STRES.EXE Module

An error has been encountered in opening the problem-name.STE file (stress file for static analysis), or the problem-name.STP file (stress file for dynamic analysis).

Delete the file and rerun the Stress module again.




Error In Opening/reading File .LCD (For Submodel Constraints)

STAR.EXE Module

The *problem-nam.LCD* file is the main displacement file which is used along with the *problem-name.LC2* file for submodeling problems. An I/O error has occurred in opening the *LCD* file.

Check your system and input.

Error In Reading Temperature File (.HTO)

STAR.EXE or STRES.EXE Module

The *problem_name.HTO* contains nodal temperatures. It is generated by thermal analysis (HSTAR or FFE Thermal). STAR and FFE Static may use this file to calculate displacements, strains, and stresses due to thermal loading specified by this file. The STAR.EXE or the STRES.EXE module has failed in reading this file.

Check your input for the TEMPRead command (Loads-BC, Load-Options, Read Temp. As Load). View temperature results. Run thermal analysis again if you can't view temperatures and then try static analysis again.

Error - Orthotropic Properties Are Not Proper (See Output)

STAR.EXE Module

Orthotropic properties are not proper for a material set. The Poisson's ratios do not obey the following rule:NUij*NUji>1, where indices i and j run from 1 to 3 and represent global directions X, Y and Z.

Check the end of the output file for error messages indicating the material set number with this problem.

Error Reading Dispersed File (To Another Drive): <Name.ext>

STAR.EXE and DSTAR.EXE Modules

The specified file corresponds to one of the partitioned files redirected for storage on other drives. The first character of the name extension specifies the type of file (S = Stiffness, G = Geometric Stuffiness/Consistent Mass and K = duplicate file for Geometric Stiffness). The second and third character constitute a two digit number specifying the partition number.

Contact SRAC if you got this message.

Error Reading .MAS File <Data>

DSTAR.EXE Module

The file with extension "MAS" is a major internal file in DSTAR.

The <data> corresponds to the information which can not be read from the file. If the <data> is displayed as MAXA, it corresponds to the offset information of the stiffness file. If <data> is displayed as AM, then it corresponds to the mass matrix data.

Error Reading Material Damping Value From File: .MSF

STAR.EXE or STRES.EXE Modules

You have specified to use material damping. The problem-name.MSF file contains the relevant information. The program found wrong information in this file indicating possible database corruption. Refer to the on-line help for the PD_DAMPREAD command (Analysis, Post Dynamic, PD_DAMP/GAT, Read Material Damp.)

Review your input and try again. Contact SRAC if the error persists.

Error - Target Element Length/area Is Zero (Bonding/CPCNS Set No. = <Number>)

STAR.EXE Module

If the specified set corresponds to a point-to-curve type of constraint, then the length of the target curve must be greater than zero. The distance between points 2 and 3 must be greater than zero as in figure D-1. If the specified set corresponds to a point-to-surface type of constraint, then the area of the target element must be greater than zero (see Figure D-2)

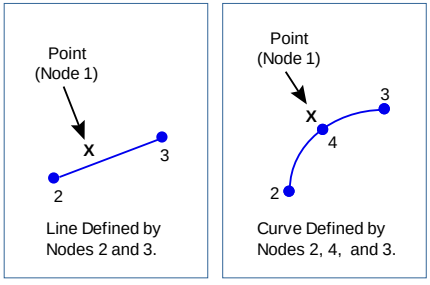
List the constraint set and check the element connectivity for possible overlapping or collapsing of the target nodes.

Error - Target Element Type Is Unacceptable (Bonding/CPCNS Set No. = <Number>)

STAR.EXE Module

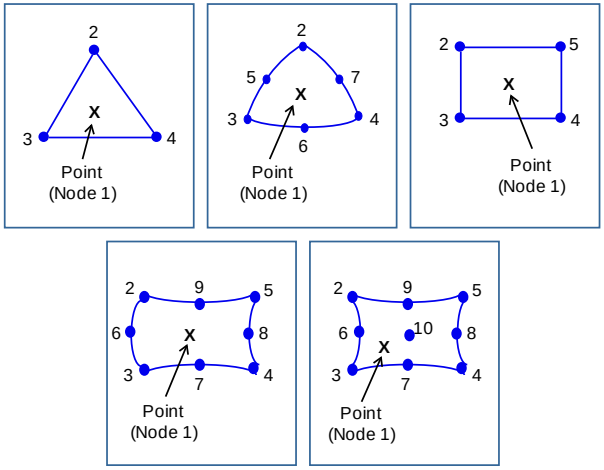
If the specified set corresponds to a point-to-curve type of constraint (or bonding), then the total number of nodes can only be 3 or 4 and should be arranged on the curve according to:

Figure D-1. Point-to-Line Constraints



If the specified set corresponds to a point-to-surface type of constraint (or bonding), the total number of nodes can only be 4, 7, 5, 9, or 10 and should be arranged on the surface according to the following convention:

Figure D-2. Point-to-Surface Constraints



Error - Unable To Find Reference Coordinates For Source Point (Boundary/CPCNS Set No. = <Number>)

STAR.EXE Module

The program uses an iterative algorithm to find the reference coordinates for the source node within the target area (element) in the case of point-to-line, point-to-surface, or bonding constraints. The error message is displayed when the reference coordinates are not found within the maximum number of iterations(=50), or when the target element may not properly defined.

Check your input and try again.

Error, Zero Or Negative Jacobian Determinant, Element: <Number>

STAR.EXE or STRES.EXE Module

The determinant of the Jacobian matrix for the specified element is equal to or less than zero.

The element may have excessive distortion or elements with zero area or volume. List and plot the element, check its connectivity and shape. You may need to remesh or redefine the element.

ERROR - Zero Volume. Element Number <Number>

STRES.EXE Module

The specified element has zero volume (or area).

List the element and verify the validity of its nodal connectivity. Examine collapsed nodes if any.

File X.BAT Not Found. Execution Aborted

RENUM.EXE, STAR.EXE or STRES.EXE Module

The X.BAT is created internally by the pre-processor. It is used to track the name of problem to be solved when various modules are used. Once all modules associated with the problem are executed, the file is used to open the pre-processor again. The STAR.EXE module could not find this file.

You should not see this error if you are running analysis from the pre-processor. If you are running the problem manually outside the pre-processor (i.e., one module at a time), make sure that this file exists after execution of the PRE1.EXE module. See note below:

For example, to run a static analysis manually: 1)create an ASCII text named X.BAT, 2) type the name of the problem (without any extension) and then close the file, 3) Close the Pre-processor, 4) Execute RENUM (if desired), 5) Execute Pre1, 6) execute STAR, and finally, 7) execute Stress (if desired).

Fluid Option Is Not Available With Non-axisymmetric Loading

STAR.EXE Module

The fluid option is not available with asymmetric loading.

Redefine the element group or use full 3D model. This version does not support the two option simultaneously.

Fluid Element Stresses Are Not Available With Random Vibration

STRES.EXE Module

You requested to perform random vibration analysis on incompressible materials.

This version does not support Fluid option (incompressible materials) for random Vibration analysis.

For Asymmetric Buckling, Only Inverse Method Is Available

STAR.EXE Module

You have used an option other than Inverse Method to evaluate the Buckling loads and mode shapes for Asymmetric buckling analysis. In this release, only the Inverse Method is available for this type of problems.

Use the A-BUCKLING command (Analysis, Frequency/Buckling, Buckling Options) to choose the Inverse Power Method and run buckling analysis again.

For Asymmetric Problems, Geometric Stiffness (Or In-plane Loading Effect) Is Calculated Only In Frequency And Buckling Analyses

STAR.EXE Module

You have included the Stress Stiffening effect (in-plane effect) for a Static analysis of an Asymmetric model. This option is not supported in this release.

Use the A-STATIC command (Analysis, Static, Static Analysis Options) to deactivate the stress-stiffening flag (in-plane effect).




For Random Vibration, Support Motion Must Be Defined In Global Cartesian Coordinate System

STAR.EXE Module

You have defined the Support Motion (for Multiple Base Motion) in a local coordinate system. This feature is supported for Time History and Harmonic analyses only. It is not supported for Response Spectra and Random Vibration analyses.

Use the Global Cartesion coordinate system to specify support motion.

For Response Spectra, Support Motion Must Be Defined In Global Cartesian Coordinate System

STAR.EXE Module

You have defined the Support Motion (for Multiple Base Motion) in a local coordinate system. This feature is supported for T*ime History* and *Harmonic analyses* only. It is not supported for Response Spectra and Random Vibration analyses.

Use the Global Cartesion coordinate system to specify support motion

Forming Stiffness (Required Shift Might Be Different Than Saved One)

STAR.EXE Module

This is only a warning message.

See the message above.

Forming Stiffness (Saved Stiffness Matrix Affected By Frequency Shift)

STAR.EXE Module

This is only a warning message. You have requested to use the decomposed stiffness matrix from the previous run instead of calculating a new one. The stored data from the previous run indicates that the previous run was computed for a frequency analysis with a frequency shift. Any non-zero shift alters the structure of stiffness matrix and makes it unusable for any future static runs (as well as future frequency runs with a shift different than the original value used to save the stiffness matrix).

The program gives this message, deletes the saved decomposed stiffness matrix and calculates a new one.

Geometric Stiffness Not Available For Elbow Element

STAR.EXE Module

The ELBOW element does not support calculation of the geometric stiffness which is needed for buckling, static, frequency problems with active in-plane stiffening effect.

Use a fine mesh of BEAM3D elements instead of each ELBOW element.

Geometry Of A 2D Element Is Reduced To A Line, Element <Number>

STAR.EXE Module

The specified 2D element is reduced to a line as its corner nodes are collapsed. Its area is zero.

List the element and check the validity of its nodes and their connectivity.

ID Array File (.IDA) Missing

STAR.EXE or STRES.EXE Module

The *problem-name.IDA* file is an internal file that stores ID Array information. STAR or STRES could not find the file. The file was not generated or was generated but then deleted.

Try again.

Ignoring The Geometric Stiffness In Mode Shape Cal.

STAR.EXE Module

This is only a warning message indicating that the in-plane stiffening is not allowed in frequency analysis of piezolectric materials.

Note the warning.

Improper First Row In .DIR File

STAR.EXE Module

The Problem-name.DIR file is created by the program to specify the storage locations for large problems where the available disk space on one directory (or disk) is not sufficient.

The first row of this file does not specify a valid directory. Refer to the Large File Management chapter in the COSMOS/M Basic System manual for details.

Improper Third Node For Beam Element Number <Number>

STAR.EXE or STRES.EXE Module

The third node is used to define the orientation of the cross section of the beam element in space. The third node may not be collinear or (almost collinear) with the other two node.

Redefine the element such that the third node is not collinear with the other two nodes.

IO-ERROR, FILE: <Extension>

DSTAR.EXE Module

This message indicates that there is some input/output error in the file specified by its extension. The files with extensions ".MSF", "IDA" or "OUT", correspond to frequency, ID array or output files, respectively.

Check your input and system and try again.

IO: ERROR, FILE:.IDA

STAR.EXE Module

The file *problem-name.IDA* file is an internal file created for buckling analysis of asymmetric problems. The error message indicates that the file already exists.

Delete or rename the file and try again.

I/O Error In Reading Displacement File .LCD

STRES.EXE Module

Displacements can not be read from the displacement file (with the extension LCD). Make sure that the file exists in the working directory.

You may try to rerun the problem from the beginning. If the error persist contact us in the SRAC to fix the problem.

I/O Error In Reading Old Displacement File

STAR.EXE Module

*Refer to the message "**Displacement/Reaction not found from old file for node <number>".***

I/O Error In Writing File .LCM

DSTAR.EXE Module

This error occurs during the storage of mode shape results in the plot file with extension "LCM".

Check your disk and try again.

I/O Error <Number> In File, <Name>

STAR.EXE Module

The problem-name.LCD file is the main displacement file and the problem-name.LC2 file is the auxiliary displacement file. The .LC2 file is used in special analysis cases like submodeling and asymmetric loading. An I/O error has occurred.

Check your system and input and then try again.

Improper Third Node For Boundary Element Number<Number>

STRES.EXE Module

The specified third node of this element is not valid.

List the element and check that the three nodes are not collinear. Redefine the element to use proper nodes and try again."

In Random Vibration , AnalysisAll The Support NodesAssociated To A Multi-baseMotion Curve, Must Have The Same Curve Multiplier

STAR.EXE Module

You have associated more than one set of Support Levels to the same curve with each set assigned a different Curve Multiplier for Random Vibration analysis.

In this release, all Support Level sets associated with a curve must be assigned the same Curve multiplier for Random Vibration Analysis.




In Response Spectra Analysis, All The Support Nodes Associated To A Multi-base Motion Curve, Should Be Excited In The Same Direction

STAR.EXE Module

You have associated more than one set of Support Levels with the same curve with each set defined in a different excitation direction for Response Spectra analysis.

In this release, all the Support Level sets associated with a curve must be defined in the same excitation direction for Response Spectra Analysis.

Inside And Outside Nodes Of Submodel Constrained By Eq. Set <Number>

RENUM.EXE Module

The nodes defining a constraint equation should be either all inside or all outside of a submodel.

Modify your input and try again.

Integration Pt. Not Inside Any Element For Point <Number>

STRES.EXE Module

For the ASME Stress Check, the specified integration point lies outside of the model and no stress results are available for it.

Use the ASMESECDEF command (Analysis, Static, ASME _CODE, Define ASME Section) to redefine the two ends of the section to avoid this situation.

Invalid Internal Analysis Flag For Basic System

STAR.EXE Module

The STAR.EXE program processes static, frequency and buckling analyses. The corresponding internal flags are 0, 2 and 1, respectively. The error message will be displayed when STAR finds a number other than 0, 1, and 2.

This is an internal error that happens when STAR is executed outside GEOSTAR. You should never see this message if you run from GEOSTAR.

Large Warping, Use Quad4 Element; Twisted Element <Number>

STAR.EXE Module

The specified element is a rectangular shell element formulated using the quadratic option (QUAD option). This formulation is essentially suitable for flat or moderately twisted elements. Its use for excessively twisted elements results in substantial inaccuracy if it is used in its original form. For moderately twisted elements, applying Warping Correction could ease the problem and yield acceptable results. This message indicates that the amount of twisting for the specified element is beyond the level that can be fixed by applying the Warping Correction.

List the element to find out its element group and then use the EGROUP command (PropSets, Element Group) to redefine the element group formulation. Use the QUAD4 or QUAD2 formulation options. You may also remesh the model with triangular elements (SHELL3/SHELL3T).




Layer Thickness Is Less Than <Value> At Layer <Number>for Element <Number>

STAR.EXE Module

The thickness of the specified layer in the specified element is either zero or too small.

List the element to find the associated real constant set and redefine it to include a valid value for the thickness.

Load Balance Is Not Available With Asymmetric Option (Load Balance Is Not Checked)

STAR.EXE Module

This is only a warning message when the Force Balance at grid nodes is requested for asymmetric problems. In this version, the balance of forces can not be calculated for asymmetric problems.

The program will continue by ignoring the request for force-balance.

Magnitudes Of Loads Are Too Small. Please Increase Them And Rerun The Program

DSTAR.EXE Module

The magnitude of the applied buckling loads are too small. Such small loads could cause numerical inaccuracy in the computation.

Increase all the applied loads by a reasonably large factor.

Mass Matrix Not Defined (You May Not Have Defined Density)

DSTAR.EXE module

The error indicates that the mass matrix is not calculated. Check to see whether you have properly defined the material density.

Master D.O.F. Equal To Zero

STAR.EXE Module

You are running a Substructuring problem with the number of master d.o.f. equal to zero.

Check your model to see if you have defined proper master nodes.

Master D.O.F. Exceed The Limit <Number>

STAR.EXE Module

You are running a Substructuring problem with the number of master d.o.f. larger than the specified limit. Refer to the end of the output file for the error message indicating the number of master d.o.f. in your problem.

You may have to reduce the number of master d.o.f. and try again.

Material Property Is Not Homogeneous Inside J-integral Path: <Number>

STRES.EXE Module

The elements inside the specified J-integral are not associated with the same material property set.

If possible, reduce the size of the path such that it lies within one material.

Material Property Is Not Isotropic Inside J-integral Path: <Number>

STRES.EXE Module

Not all elements inside the specified J-integral have isotropic material properties.

Redefine the path such that it is contained within one isotropic material

Missing File, Return To Preprocessor. <File.IDA>

STAR.EXE Module

The *problem_name.IDA* file is missing. This file contains ID Array information and should have been generated by PRE1.EXE and then used by STAR.EXE. The above error message occurred during the reading of this file by STAR.EXE. PRE1.EXE has not generated the file or the file was deleted afterwards. A corrupt database may result in this message also.

Check your data and rerun the problem. You may have to regenerate a new database for this problem by loading the session file to a new problem.

Missing Files. Run The Pre-stress Analysis First

STRES.EXE Module

This message indicates that the pre-stress analysis has not been completed successfully prior to running the Stress module. Depending on the type of the analysis, the pre-stress analysis could be static displacement calculation (the STAR.EXE module), mode shapes (the DSTAR.EXE module), or dynamic displacements response (DPOST.EXE).

Please run the appropriate pre-stress analysis first and then run the stress module if needed (running the static module includes running the stress module by default).

Modal Acceleration Method Not Available With Consistent Mass

STAR.EXE Module

You requested frequencies using the consistent mass matrix formulation and you are trying to run time-history analysis with the Modal Acceleration Method (MAM) option. DSTAR supports this option only if lumped matrix formulation was used to calculate frequencies in DSTAR.

Use the A-FFEQUENCY command (Analysis, Frequency/Buckling, Frequency Options) to specify lumped mass formulation and try again.

Mode Acceler. Method Is Not Available With Consistent Mass

DSTAR.EXE Module

You can not run a frequency analysis with both the consistent mass and mode acceleration flags activated simultaneously.

You may use the A-FREQUENCY command (Analysis, Frequency/ Buckling, Frequency Options) to switch to the lumped mass formulation and try again.




Modulus Of Elasticity (Ex) Smaller Than 1.e-25 For Element <Number>. (Check Material Group <Number>)

STAR.EXE Module

The modulus of elasticity (EX) for the specified element is smaller than 1.e-25.

Check the specified material group using the MPLIST command (PropSets, List Material Props). Use the PropSets menu to pick a material from the library or define an acceptable value for EX, and try again.

MORE THAN <Number> Layers Are Used For A PLATE-SHELL Element <Number>

STRES.EXE Module

The number of layers is larger than the specified limit for the specified element.

Reduce the number of layers and rerun the problem all over again.

Multi-base Motion Is Not Available With Consistent Mass

DSTAR.EXE Module

You can not run a frequency analysis with both the consistent mass and multi-base motion flags activated simultaneously.

You may use the A-FREQUENCY command (Analysis, Frequency/ Buckling, Frequency Options) to switch to the lumped mass formulation and try again.

No Gravity/centrifugal/ inertia Force Is Calculated (Material Density May Not Have Been Defined)

STAR.EXE Module

You have activated the gravity and/or centrifugal flag(s) and the program is not able to calculate the equivalent forces. The equivalent forces are computed by multiplying mass by acceleration.

Verify that a non-zero mass density is defined and check your input for the proper load case including acceleration and velocity values.

No Load Or Prescribed Displacement Is Defined (For Problems, With Special Loads Such As Thermal, Gravity and Centrifugal Activate The Proper Flag In A_STATIC Command)

STAR.EXE Module

No loads were specified. You must specify some type of loading, otherwise all results are zeros. The external load could be in the form of mechanical loads (such as forces, pressures), thermal loads (such as nodal temperatures and thermal gradient on shell and beam elements), and Gravity or Centrifugal loads (due to mass inertia). This message may obtained while running static, buckling, or frequency analysis with active in-plane loading. Only the active load case is considered for buckling problems and static and frequency problems with active in-plane loading flag.

The user-defined mechanical loads are always considered in the problem solution. However, to consider other types of loading, you must activate the corresponding flag in the analysis options command. For gravity and centrifugal loading mass density must be specified. For thermal loading the

coefficient of expansion must be specified. Also make sure that the proper load case is activated. Use the A_STATIC (Analysis, Static, Static Analysis Options) command to activate the proper flag(s).

No Load Or Prescribed Displacement is Defined For ACTIVE LOADCASE (For Problems, With Special Loads Such As Thermal, Gravity and Centrifugal Activate The Proper Flag In A_STATIC Command)

STAR.EXE Module

No loads were specified.

For details, refer to the previous message "No Load or prescribed displacement is defined (for problems, with special loads such as Thermal, Gravity and Centrifugal activate the proper flag in the A_STATIC command).

No. Of Interface Elements (In Main Model Side) Is Larger Than <Value>

STAR.EXE Module

The submodel is very large. The number of elements along the submodel interface is larger than the specified limit.

Reduce the size of the submodel and rerun the problem.

No Thermal Load Application With Piezoelectric Materials

STAR.EXE Module

The piezoelectric element (SOLIDPZ) does not support thermal load application.

Deactivate thermal loading and try again.

Non-existent Node For Element <Number> Node: <Number>

STRES.EXE Module

This message indicates that the specified element includes a node which is not available.

List the element and redefine it to use proper nodes that are available in the database.

Not Enough Memory To Run; Extra Memory Needed <Unit> <Size>

STAR.EXE, DSTAR.EXE or STRES.EXE Modules

In order to run this particular problem you need to have the specified extra amount of free memory in the specified unit. The required memory increases as the size of the problem increases.

Increase the size of the memory available by closing other applications. If the amount of RAM on your machine is small, you may need to install extra RAM on your computer.




Not Enough Memory To Run RENUM

RENUM.EXE Module

There is no enough memory to run this problem.

Increase the size of the memory available by closing other applications and rerun the problem. You may also provide the needed memory by using the virtual memory. To do so, use the command RENUM command (Analysis, Renumber). If the size of RAM on your machine is small, we suggest to install additional RAM.

Not Enough Memory To Run The Problem

DSTAR.EXE Module for Guyan Reduction

In order to run this problem, you need to increase the amount of free memory.

Increase the size of the memory available by closing other applications. If the size of the RAM on your machine is small, you may increase the memory by installing extra RAM.

Not Enough RAM For Requested Memory To Store Stiffness Matrix. Available RAM Is: (Mega Bytes) = <Size>

STAR.EXE Module

The specified size is the available size of memory which is smaller than the size requested by the user.

Reduce the requested size to the specified size. You may increase the size of the memory available by closing other applications or installing extra RAM.

NOTE: Sum Of The Free Disk Spaces Of Participating Drives Is Smaller That The Required Space For This Problem: Total free disk spaces (<unit>) = <size> Required space (<unit>) = <size>

STAR.EXE Module

The total free disk space in all drives specified by the Large File Management utility is not sufficient to solve this problem.

In the absence of extra disk space, you may have to reduce the size of the problem by using larger elements. Sometimes using the other option in the RENUM program may reduce the stiffness bandwidth, and consequently the total size of the problem.

Options 1 And 2 Of File Transfer To Other Directories Not Available With Piezoelectric. You May Use Option 3 Instead

STAR.EXE Module

For this piezoelectric problem, you have requested to partition the stiffness file and store each part in a separate directory (options 1 and 2 of Large File Management). These two options are not available in this version for the piezoelectric problems.

You may use option 3 to transfer the whole stiffness matrix (or similar files) to directories other than the working directory.




Options 1 And 2 Of File Transfer To Other Directories Not Available With Substructuring. You May Use Option 3 Instead

STAR.EXE Module

For this substructuring problem, you have requested to partition the stiffness matrix and store each part in a separate directory (options 1 or 2 of Large File Management). These two options are not available in this version for substructuring problems.

You may use option 3 to transfer the whole stiffness matrix (or similar files) to directories other than the working directory.

P-method Is Not Available With Asymmetric Option

STAR.EXE or STRES.EXE Module

The asymmetric option is only available with the H-Method. You have specified the P- or the HP-Adaptive Method for this Asymmetric problem.

Use the H-Method and try again.

Previous Run Is Done With Diff. Option In .DIR File, Thus Saved Stiffness Matrix Can Not Be Retrieved

STAR.EXE Module

You have requested the use of the stiffness matrix from a previous run which used a Large File Management option to store the results. The option you are using now is different than the one that you have used when you saved the stiffness matrix. Due to this inconsistency, the current run failed.

Use the proper option number and rerun the problem.

Problem Is Too Large For The Subspace Method. Number Of Eigenvalues Have Been Reduced To <Number>

DSTAR.EXE Module

This warning message only is displayed for the problems with Consistent Mass option when the problem is too large for the available memory.

If you still wish to solve the problem with the original options:

a. Increase the size of available memory, or,

b. Reduce the number of D.O.F., or,

c. Reduce the number of requested frequencies.

Problem Too Large For The Program

DSTAR.EXE Module

It indicates that the number of requested frequencies is larger than limit for the Lanczos method.

Either reduce the number of frequencies or use another method such as Subspace and rerun the problem.




QM6 Option Not Available With Asymmetric Loading, Full Integration Used

STAR.EXE Module

This is only a warning message. The QM6 option of computing the stiffness matrix is not available for asymmetric PLANE2D elements.

The full integration option will be used instead.

Reactions Are Not Available With Asymmetric Option (Reaction Is Ignored)

STAR.EXE Module

This is only a warning message in asymmetric problems when the Reaction flag is on. In this version of COSMOS/M the reaction force can not be calculated at the constrained nodes.

The program will continue by ignoring the request to calculate reaction forces.

Replace PREVIOUS Run Results With CURRENT Iteration (Y/N)?

DSTAR.EXE Module

You may get this message if you press the Esc key during the rerun of the DSTAR.EXE program when results are available from a previous run.

If you press Y then the program will complete the current iteration and use it to replace the results of the previous run. The completion time depends on the size of the problem as well as the stage of calculation at which you have pressed the ESC key. If you press N, the computation will be terminated and the previous results will be retained for postprocessing. Note that even if you initially press Y, you can still terminate the program at any time by pressing the ESC key again.

Request For Program Termination Press Esc To Terminate Or Any Other Key To Continue

RENUM.EXE, STAR.EXE, DSTAR.EXE or STRES.EXE Modules

You have pressed the *Esc* key intentionally or accidentally.

Pressing of the Esc key interrupts the module being executed. Press the Esc key again to terminate or press any other key to cancel the interruption and continue running.

Requested Block Size Is Smaller Than Minimum <Size> <Unit>

STAR.EXE Module

The specified size is the minimum size required for an internal storage array in order to run this particular problem. The required memory increases as the size of the problem increases. This internal array is used to store two blocks of the stiffness matrix as well as other information at different stages of the run. These other tasks require the specified minimum size of memory in the prescribed unit (i.e., byte, kilo byte, mega byte).

Increase the size of the memory available by closing other applications or add more memory to your computer to improve the performance.

Same Base Excitation Is Defined In More Than One Support Level

STAR.EXE Module

Multi-base excitations at a certain node in the same global Cartesian direction are referred to in more than one support level. This results in a node being assigned two different support levels in two different local coordinate systems which may have non-zero resultant component in the same direction of the global cartesian system) which is not supported in this release. The node label and corresponding direction is specified in the output file (*problem-name.OUT*).

Modify your input and try again.

Security Device Is Not Responding. Please Install Security Device And Type Any Key To Continue. Type Esc Key To Quit From The Program

STAR.EXE or DSTAR.EXE Module

Check your license scheme. If you are using a security lock, make sure that it is properly connected and the proper license file exists in the COSMOSM directory.

If you still can not execute the program please contact SRAC.

Singularity In Computation Of Average Plane For Warped Element <Number>

STAR.EXE Module

The program could not apply Warping Correction for the specified element defined with the QUAD option.

List the element to find out its element group and then use the EGROUP command (PropSets, Element Group) to redefine the element group formulation. Use the QUAD4 or QUAD2 formulation options. You may also remesh the model with triangular elements (SHELL3/SHELL3T).

Spin Softening Is Not Available With Consistent Mass

STAR.EXE Module

You have activated the spin softening flag and specified consistent mass formulation. These two options are not compatible in this release of COSMOS/M.

Use the A-FFEQUENCY command (Analysis, Frequency/Buckling, Frequency Options) to specify lumped mass formulation or deactivate the Spin Softening flag.

Static Analysis Data Not Available For The Dead Load Case <Number>

STRES.EXE Module After The Advanced Dynamic Analysis (The DPOST.EXE Module).

You have requested to use the results of a static load case. The results for the specified load case were not found.

Make sure that you have run the static analysis (including the stress calculation) for this load case prior to running the advanced dynamics.

STOP - ASME Stress Check Is Only Available When Stress Is Requested In Global Cartesian Coordinate System

STRES.EXE Module

Stop ASME stress check is only available when stress is requested in Global Cartesian Coordinate system.

Change the option for stress coordinate system in the element group command to global and run the stress module again (no need to run for displacement again).

Stop - Asymmetric Loading Is Used With More Than 1 Load Case

STAR.EXE or STRES.EXE Module

Multiple load cases are activated with the asymmetric option. Only one load case may be activated.

Use the LCSET (Analysis, Static, Activate Load Case) command to make sure that only one load case is active for analysis.

Stop - Consistent Mass is Not Available With SHELL6 (Switch to the Lump Mass Option)

STAR.EXE Module

Consistent Mass formulation is not available for SHELL6 elements.

Use the analysis command (Analysis, Frequency/Buckling, Frequency Options) to specify the lumped mass option. You may instead remesh your model with SHELL3T or SHELL9 elements and keep the consistent mass formulation option.

Stop - Consistent Mass Is Not Available With SHELL9 (Switch To The Lump Mass Option)

STAR.EXE Module

The 8/9 node isoparametric shell element (SHELL9) does not support the consistent mass option.

Use the analysis command (Analysis, Frequency/Buckling, Frequency Options) to specify the lumped mass option. You may also replace the SHELL9 element with a lower order element such as SHELL4T which supports the consistent mass option.

Stop - Consistent Mass Is Not Available With SHELL9L (Switch To The Lump Mass Option)

STAR.EXE Module

The 8/9 node isoparametric layered shell element (SHELL9L) does not support the consistent mass option.

Use the analysis command (Analysis, Frequency/Buckling, Frequency Options) to specify the lumped mass option. You may also replace the SHELL9 element with a lower order element such as SHELL4T which supports the consistent mass option.




Stop - Determinant of Jacobi Equal To Zero For Element <Number>

STAR.EXE Module

The determinant of the Jacobian matrix for the specified element is equal to or less than zero.

The element may have excessively distorted elements or elements with zero area or volume. List and plot the element, check its connectivity and shape. You may need to remesh or redefine the element.

Stop - D.O.F. for Super Nodes Did Not Match. At Supper Element <Number>

STAR.EXE Module

The number of degree of freedom for the specified superelement does not match between the two internal files.

The database may be corrupted. Start a new problem, load the session file using File-Load and try again.

STOP - Element Area Is Smaller Than Tolerance <Value>, Element <Number>

STAR.EXE Module

The surface area of the specified element is smaller than the tolerance value. Elements smaller than specified value cause numerical difficulty in the computation and should be avoided.

Check the following:

1.Check the mesh to make sure that all the nodes of the specified element are not collinear.

2.Check to see whether this problem is happening only for a few elements, you may be able to modify the mesh locally to produce larger size element(s).

3.If many elements have this problem, then may be the overall dimensions of the model is very small, you may regenerate your model using a smaller unit of length, and remesh the model. In this case, you also need to modify your material properties accordingly.

4.In case you have a very fine mesh with a lot of elements, check the possibility of meshing with a larger element size without compromising the accuracy of the solution.

Stop - Error In Reading Displacement File (.LC1)

STRES.EXE Module

The *problem-name.LC1* stores displacements for asymmetric problems. An error was encountered in reading this file.

Rerun the problem again to calculate displacements and stresses. Contact SRAC if the problem persists.

Stop - Error In The Element Connectivity At Element <Number> Please Check The Output File For Detailed Information.

STAR.EXE Module

The specified element has improper connectivity.

List the element and check its nodal connectivity and associated element group. The output file provides the coordinates of the associated nodes for your review. If you created the element manually (not through meshing), check the order of the nodes.

Stop - EX Is Equal To Zero For Element: <Number>

STAR.EXE Module

Modulus of elasticity (EX) is not defined or is smaller than 1.e-20 for the specified element.

List the element to find out the associated material set. Use the PropSets menu to define a proper value of EX for the specified material set.

Stop - File .DIA Not Available

DSTAR.EXE Module

The problem-name.DIA file contains the diagonal of the stiffness matrix.

Make sure that the STAR.EXE module was completed successfully prior to running the DSTAR.EXE module. If you are running the modules interactively (i.e., one by one outside GEOSTAR), make sure that you did not skip the STAR.EXE module.

STOP - For Tetra, Polynomial Order <Number> Is Not Available Yet

STAR.EXE Module

The specified order for P-method is not available for Tetrahedral elements in this version. The maximum order for this type of element is 6 at present.

Use a polynomial order of 6 or smaller for Tetrahedral elements.

STOP - Improper Crack. See Output File

STAR.EXE Module

For a 3D-Crack element the three nodes defining the geometry of the crack are collinear.

Check the end of the output file for error messages to find the number of the crack element which has this problem. List the crack element and the associated solid element and check the nodal connectivity. Fix the error and try again.

Stop - Improper Real Constant Set Is Used For Beam Element <Number>

STAR.EXE and STRES.EXE Module

The entries in the real constant set associated with this beam element are invalid.

List the element to find out the associated real constant set and then list the real constant and examine its entries. Fix the problem by redefining the real constant set or associating the element with a valid real constant set.




Stop - Invalid Element Type With Asymmetric Loading

STAR.EXE or STRES.EXE Module

You have activated the Asymmetric option in your problem and your model contains element types that are not supported by the asymmetric option. The supported element types are: axisymmetric shell elements (AXISHELL) and rectangular plane elements (PLANE2D) only.

Remove or replace the incompatible elements and try again.

STOP - Invalid PLANE2D Element Option For Asymmetric Loading

STAR.EXE or STRES.EXE Module

You have activated the asymmetric option in your problem but a wrong option was specified for a PLANE2D element group. Available options of PLAN2D elements are: Plane Stress, Axisymmetric, Plane Strain and Asymmetric.

Use the EGROUP command (Propsets, Element Group) to select the Asymmetric option for PLANE2D element groups and try again.

STOP: J-integral Is Not Available With Centrifugal Loads

STRES.EXE Module

Body forces such as centrifugal loads are not supported in this version of COSMOS/M for the J-integral consideration.

The only valid types of loads are concentrated loads, pressures and Thermal loads.

STOP: J-integral Is Not Available With Gravitational Loads

STRES.EXE Module

Body forces such as gravitational loads are not supported in this version of COSMOS/M for the J-integral consideration.

The only valid types of loads are concentrated loads, pressures and thermal loads.

STOP: J-integral Is Not Available With P-Methods

STRES.EXE Module

You are using the J-Integral with a P-method. The J-integral is available only with the H-methods. The valid elements are the PLANE2D and TRIANG types.

Change the P-method to H-method and run the problem from the beginning (including the displacement calculations).




Stop - KG Is Not Available For The SOLIDL Element

STAR.EXE Module

The KG (Geometric Stiffness) is evaluated for buckling problems and linear static or frequency problems with active in-plane loading effects. Geometric stiffness is not calculated for SOLIDL elements. Therefore, you can not use this element for such problems.

You may be able to use composite shell elements like SHELL3L, SHELL4L, or SHELL9L instead. Otherwise, remodel your geometry and use regular solid elements.

STOP - Maximum Node Number Exceeds The Limits

STAR.EXE Module

The number of nodes in your model is larger than the maximum allowable number.

Start a version that supports more nodes (the 128K or the 256K node/ element versions in your COSMOSM program group). Or remesh with a larger element size.

Stop - Missing File <Name.ext>

STRES.EXE Module

The specified file could have one of the three extensions: “DIS”, “EIG” or “IDA”. The “DIS” corresponds to the internal file generated during the displacement calculation. If this file is missing, then most likely you have not run STAR to calculate displacements. The “EIG” extension corresponds to the internal file for the frequency analysis and “IDA” corresponds to the file storing the ID Array (relating the nodal d.o.f’s to the global d.o.f.’s).

Check your data and run static analysis again (displacements and stresses).

STOP - Modal RMS Disp. File Does Not Exit

STRES.EXE Module

The file including the modal displacement response from Random Vibration analysis is not available.

Make sure to perform Random Vibration analysis successfully prior to running the stress module.

Stop - Moment Of Inertia, <Component> Is Equal To Zero <element>

STRES.EXE Module

The specified component of cross-sectional inertia was not defined or was defined with a value of zero for the specified element.

Inertia component could be either IY or IZ. List the Real Constant sets, define the missing components and then run full static analysis (displacements and stresses).




Stop, More Than <Number> Layers Are Used For A Shell9l Element <Number>

STAR.EXE Module

The number of layers in the specified element exceeded the limit.

List the element and redefine its real constant set. If you need more layers than the limit, consider remodeling your geometry and use multiple layers of elements.

STOP - NBLK2 Exceed The Limit <Number> NBLK2 = <Number>

STAR.EXE Module

It indicates that the number of stiffness blocks (NBLK2) is larger than the specified limit for a piezoelectric problem. The problem size is too large.

Reduce the size of the problem or increase the size of memory. Increasing the size of memory reduces the number of blocks needed to solve the problem.

Stop - Node File Is Not Available

RENUM.EXE Module

The absence of the node file indicates that the mesh has not been generated prior to running of the analysis.

Make sure that the FEA model is properly defined before running analysis.

Stop - Number Of Blocks Exceed The Limit: <Number>

STAR.EXE Module

The number of blocks needed to store the stiffness matrix exceeds the specified limit.

Increase the size of each block by freeing more memory or adding more RAM. If you used the Large File Management utility and had intentionally reduced the size of the block, then you may want to increase it and try again.

Stop - Old Shift Is Diff. Than The New One. Run The Problem From The Beginning (Including The NSTAR Run)

DSTAR.EXE Module

You have requested the use of the stiffness matrix generated by the nonlinear analysis module NSTAR. The flag is controlled by the A_FREQUENCY command (Analysis, Frequency/Buckling, Frequency Options) or the A_BUCKLING command (Analysis, Frequency/Buckling, Buckling Options). If you apply a frequency shift in your first run, thus altering the stiffness matrix, then you can not rerun DSTAR for a second time using a different shift value unless you rerun the NSTAR to update the stiffness matrix prior to running DSTAR

Use the same shift as in the first analysis, or run NSTAR and DSTAR again with the desired shift value.




STOP - PRE1 Program Has Not Completed Successfully

STAR.EXE Module

The STAR.EXE program is one of several modules executed for Static, Frequency and Buckling problems. In performing static analysis, the following modules are executed sequentially:

RENUM.EXE, PRE1.EXE, STAR.EXE, and STRES.EXE

In performing frequency and buckling problems the following modules are executed sequentially:

RENUM.EXE (optional), PRE1.EXE, STAR.EXE, DSTAR.EXE and STRES.EXE

If the PRE1.EXE module is not successfully completed, then the above error message will be displayed by STAR.EXE

Check the output file (problem-name.OUT) for possible messages from PRE1. Check and fix the error. Also check your disk space.

Stop - Problem Is Too Large For The Program (Lanczos)

DSTAR.EXE Module

The problem is too large.

Either reduce the number of frequencies or increase the size of available memory.

STOP - Problem Too Large For The <Type> Version. Number Of Nodes For This Problem: <Number>. Limit on the number of nodes (for this version): <number>

STAR.EXE Module

The number of the nodes in this problem is larger than the limit for this version of COSMOS/M.

GEOSTAR versions that support 64,000, 128,000, and 256,000 nodes/ elements are available in your COSMOSM directory. If you are not using the 256,000 nodes/elements version, make sure to use it from the COSMOSM program group. Otherwise contact SRAC.

Stop: Requested Harm. Type (COS/SIN) For Stress Calcul. Is Not The Same As Originally Considered In Disp. Calcul.

STRES.EXE Module

There are three options for asymmetric problems: Sine terms, Cosine terms and combination of Sine and Cosine terms. Once you calculate the displacements using one of these options, you can not switch to a different option for calculating stresses.

Use the A_ASYM command (Analysis, Static, Asymmetric Load Options) to specify the use of the same option you used in calculating displacements and then run STRESS again.




STOP - Singular Matrix For SHELL4L/SHELL3L Element <Number>, Layer <Number>

STAR.EXE Module

The stiffness matrix for these elements is not positive definite and hence not valid.

List the element to find out the associated real constant set and then list the real constant set to find the material set for the specified layer. List the material property set using the MPLIST command (PropSets, List Material Props) and check the validity of the elastic properties. Specify proper valid entries and rerun the problem.

Stop - Stiffness File (.udf) Not Available

DSTAR.EXE Module

You have requested the use of the stiffness matrix generated by the nonlinear analysis module NSTAR for frequency or buckling analysis. The flag is controlled by the A_FREQUENCY command (Analysis, Frequency/Buckling, Frequency Options) or the A_BUCKLING command (Analysis, Frequency/Buckling, Buckling Options). The *problem-name.UDF* file which contains the stiffness matrix generated by NSTAR was not found.

Verify that nonlinear analysis has been performed successfully on the same problem or deactivate the corresponding flag in the A_FREQUENCY command (Analysis, Frequency/Buckling, Frequency Options) or the A_BUCKLING command (Analysis, Frequency/Buckling, Buckling Options).

Stop - Stiffness Matrix Is Singular At Equation <Number> Node = <Number> Direction = <Number> Check The Troubleshooting Section Of The Manual.

STAR.EXE or DSTAR.EXE Module

Your stiffness matrix is singular (has zero or negative diagonal) at the specified equation (d.o.f.).

Check the following possibilities:

1. *You may have unattached node(s) or undefined element properties.*
2. *You may need more constraints or you may want to turn on the soft-spring option in the relevant analysis option command.*
3. *You may have run out of disk space.*
4. *You may have assigned incompatible orthotropic material properties.*
5. *You may have activated in-plane loading and specified excessive loading which resulted in large displacements.*
6. *You may have excessive spin softening if the corresponding flag is on (i.e., Angular velocity may be too large.)*

The value of the diagonal term causing the singularity is recorded in the output file (problem-name.OUT).




STOP - Stress Not Available In Old File For Loadcase <Number>

STRES.EXE Module

The old file corresponds to the stress plot file from the previous run. The present submodeling analysis can not read the stress results for the specified loadcase in that file.

You may not have previously run the problem for the specified load case.

STOP - Submodeling Is Not Available With Asymmetric Option

STAR.EXE Module

You have activated submodeling for an asymmetric problem. The two options may not be activated simultaneously.

Deactivate submodeling or the asymmetric option. This version does not support these two options together.

STOP - Submodeling Not Available With Substructuring Option

STAR.EXE Module

You have activated submodeling and substructuring for this problem. The two options may not be activated simultaneously.

Deactivate submoldeling or substructuring. This version does not support these two options together.

Stop - Temperature Is Greater Than Defined In The Curve

STAR.EXE Module

The temperature curve does not cover the upper range of temperatures in your model.

List your temperature curve making sure that it covers the full temperature range.

Stop - Temperature Is Smaller Than Defined In The Curve

STAR.EXE Module

The temperature curve does not cover the lower range of temperatures in your model.

List your temperature curve making sure that it covers the full temperature range.

Stop - The Latest Requested Harmonics May Not Be Included In Originally Requested Harmonics For Which Disp. Is Calculated

STRES.EXE Module

After calculating the displacements using a number of harmonics for asymmetric problems, you may speed up the stress calculation by running stress only while considering a number of harmonics that is smaller the number used in displacement calculations. The range of the harmonics in the stress calculation, however, should be within the range considered during displacement calculations. Displacement results must be available for each harmonics considered in stress calculations.

Modify your input using the A_ASYM command (Analysis, Static, Asymmetric Load Options).

STOP - Thermal Option Is Not Supported With AXSHELL Element

STAR.EXE Module

The axisymmetric shell element (SHELLAX) does not support thermal loads.

You may replace the SHELLAX element with the PLANE2D (axisymmetric option) element for which thermal loading is supported.

Stop - Thickness Equal To Zero For Pipe Element <Number>

STRES.EXE Module

The thickness is not defined for the specified PIPE element.

List the element to find out the associated real constant set, redefine the set by entering a valid value for the thickness, and run full static analysis (displacements and stresses).

Stop- Use A_STRESS To Specify The Layer Number For XY-PLOT

STRES.EXE Module

For dynamic problems modeled with multiple-layered elements, stress graphs (xy-plots) are available only for a user-specified single layer.

Use the A_STRESS command (Analysis, Static, Stress Analysis Options) to specify the desired layer and rerun the stress module.

STOP - Zero Crack Length, See Output File

STAR.EXE Module

The length of a 3D CRACK element is smaller than the allowable limit of 1.0E-6. The three crack nodes may not represent a face of a solid element or they are not associated properly with the proper element.

Check the end of the output file for error messages to find the number of the crack element which has this problem. List the crack element and associated solid element and check their nodal connectivity. Fix the error and try again.

Submodeling Not Available With Frequency /Buckling Analyses

STAR.EXE Module

Submodeling is only available for static analysis.

Deactivate Submodeling prior to running frequency or buckling analysis.

Temperature Dependent Materials Not Available With TETRA4R Use TETRA4 Or TETRA10 Instead

STAR.EXE Module

You have defined temperature-dependent material properties for a material set which is associated with the TETRA4R element type. This version of COSMOS/M does not support temperature-dependent material properties for TETRA4R elements.

You may use TETRA4 or TETRA10 elements instead of TETRA4R elements.

TETRA Stress for Random Vibration in Global Cartesian Coord. Only

STRES.EXE Module

You have requested stresses for Random Vibration analysis to be calculated in a local coordinate system. For Tetrahedral elements, stresses may only be requested in the global Cartesian coordinate system for Random Vibration analysis.

Switch the flag from local to global in the corresponding element group command and rerun the stress module again (no need to run the pre-stress analysis again). You may request the stresses in the desired coordinate system during postprocessing.

Thermal Expansion Coefficient (ALPX) is Smaller Than 1.0e-30 for Element <Number>. Check Material Group <Number>

STAR.EXE Module

The coefficient of thermal expansion is too small. It must be larger than 1.0E-30.

Check the specified material group using the MPLIST command (PropSets, List Material Props). Use the PropSets menu to pick a material from the library or define an acceptable value for the coefficient of thermal expansion ALPXEX, and try again.

Thermal Flag Is Not Consistent With The One Considered During The Displacement Calculation

STRES.EXE Module

It indicates that the thermal loading flag (in Analysis, Static, Static Analysis Options) has been changed (activated or deactivated) after calculating displacements and prior to running the stress analysis. This results in inaccurate results.

Change the flag to what it was originally and rerun the stress module (STRES.EXE) only (Analysis, Static, Run Stress Analysis). If the status of the flag is not the desired one, set as desired and run complete static analysis again.

Total No. Of Integ. Pts. Along All J-paths Larger Than Limit <Number>

STRES.EXE Module

The total number of integration points along all J-integral paths is larger than the specified limit.

Use the Analysis, J_integral menu to reduce the number of integration points and run STRES.EXE module.

Total Number Of Integration Paths For J-integral Larger Than <Number>

STRES.EXE Module

The number of J-integral paths is larger than the limit.

Use the Analysis, J_integral menu to reduce the paths to be within the specified limit and rerun the STRES.EXE module.

Two Gaps Are Sharing The Same Node (See Output File)

STAR.EXE Module

This message indicates that a node is shared by two gaps which is not supported in this version of COSMOS/M. The two gaps and the sharing node are specified in the output file.

You may be able to connect one of the two gaps to a different node without changing the behavior of the problem significantly (you may be able to change the mesh locally for that purpose).

Unable To Open Temperature File (.HTO)

STAR.EXE or STRES.EXE Module

The *problem_name.HTO* contains nodal temperatures. It is generated by thermal analysis (HSTAR or FFE Thermal). STAR and FFE Static may use this file to calculate displacements, strains, and stresses due to thermal loading specified by this file. STAR could not open this file. The file was not generated or was deleted or modified after generation.

Run thermal analysis. Check your input for the TEMPRead command (Loads-BC, Load-Options, Read Temp. As Load). View temperature results. and run static analysis again.

Unable To Read Temperature File (.HTO)

STAR.EXE or STRES.EXE Module

See the: “Error in reading temperature file (.HTO)” message.

Check your input for the TEMPRead command (Loads-BC, Load-Options, Read Temp. As Load) making sure that to ask for loading of an existing time step. View or list the results of thermal analysis and then try static analysis again.

Unable To Read Temperature File Records (.HTO)

STAR.EXE Module

The file exists but STAR is unable to read the desired records. See the: “Error in reading temperature file (.HTO)” message above.

Check your input for the TEMPRead command (Loads-BC, Load-Options, Read Temp. As Load) making sure that to ask for loading an existing time step. View or list the results of thermal analysis and then try static analysis again.

Undefined Coordinate System Used To Apply CPDOF Command At Node: <Number>

STAR.EXE or STRES.EXE Module

The local coordinate system used to specify coupling at the specified node is not defined.

List the coupling set to find the undefined coordinate system, define the coordinate system and try again.

Unsuitable Element For ASME Check (Use PLANE2D Or TRIANG)

STRES.EXE Module

Only the axisymmetric PLANE2D and TRIANG elements support the ASME Stress Check.

Deactivate the ASME Stress Check flag or use one of the supported elements. If you need to modify the type of element, make sure to run full static analysis on the problem (displacements and stresses).

Wall Thickness Is Equal To Outside Diameter For Elbow Element <Number>

STAR.EXE Module

The specified ELBOW element is invalid because the thickness of the element is equal to the outside diameter making the inside diameter zero.

List the element to find out the associated real constant set and then modify the real constant set by inputting acceptable parameters.

Warning-assigned Penalty Term For Stiffness For Local Constraint May Not Be Large Enough To Ensure Proper Results, At Node <Node>, Direction <Number>

STAR.EXE Module

This is only a waring message. The value assigned for the penalty function applicable for local constraints may not be large enough to ensure proper results. The Maximum violation of the constraints occurred at the specified degree of freedom and it is larger than 1/200 of the maximum deformation in the structure.

You may improve the results by increasing the Penalty value (stiffness constant) in the A_STATIC command (Analysis, Static, Static Analysis Options).

Warning-coupled Nodes Outside Of Submodel Ignored, Set <Number>

STAR.EXE Module

This is only a warning message. It indicates that some of the nodes associated with the specified coupling set are outside of the submodel. During the submodeling analysis, the only nodes that will be considered in the coupling are those which are part of the submodel.

List the coupling sets and verify that your input is valid.

Warning - D Matrix Is Not Positive Definite For Element <Number>

STAR.EXE and STRES.EXE Module

The Elastic Matrix which relates the Stress tensor to the Strain tensor is not positive definite for the specified element.

List the element to find out the associated material set, and then check the values of the material properties such as moduli of elasticity, Poisson's ratio, and shear moduli.




Warning - In Substructuring No. Of Reactions Are Limited To: <Number>

STAR.EXE Module

This is only a warning message. You are running a Substructuring problem with the number of support reaction d.o.f. larger than the specified limit for such problems.

The program calculates only the maximum number of reactions as specified be the given number.

Warning- Material Strengths Not Defined For Failure Analysis

STRES.EXE Module

For a layered element the Failure analysis could not be performed because the strength of the material is not defined.

The program will ignore the request for Failure Analysis and compute stresses as usual.

WARNING! Non-existent Drive

STAR.EXE Module

One of the requested drives specified by the SELDIR (Control, Miscellaneous, Select Dir for Partition) command, for use with the Large File Management utility, does not exist on your machine.

Correct the error by using the SELDIR command again.

Warning - Sandwich Formulation Ignored For Anisotropic Plate

STAR.EXE or STRES.EXE Module

The sandwich formulation option specified in the element group will be ignored if you provide the elastic matrix directly. If the matrix is not directly provided, the program will use the specified formulation to calculate the stiffness matrix as usual.

No action is needed.

WARNING: Steady State Thermal Solution Is Not Associated To Any Load Case (You May Use TEMPREAD Command To Do So)

STAR.EXE Module

This is only a warning message. Thermal loading was activated but it will be ignored because thermal loading from steady thermal analysis is not available for any active load case.

Use the TEMPREAD command (Loads-BC, Load-Options, Read Temp As Load) to assign a temperature profile from thermal analysis to a load case.

Warning, Sturm Sequence Check Can Not Be Performed For Negative Shift

DSTAR.EXE Module

The program ignores the request of Sturm Sequence Check and terminates after calculating and storing the frequencies and mode shapes.

You may correct the shift and rerun the problem.

Warning: Undefined GXZ Is Equated To GXY For material set: <Number>

STAR.EXE or STRES.EXE Module

This is only a warning message. For the specified orthotropic material set, the undefined Shear Modulus GXZ is set equal to GXY. If the GXY itself is not defined for the specified set, then: the following equation is used:

GXZ=GXY=(EX*EY)/(EX+EY+2*NUXY*EY), where EX and EY are the modulus of elasticity in the X and Y directions, and NUXY is the poisson's ratio.

Since the set numbers are displayed one after another (for the sets with undefined GXZ), thus overwriting the previous set number. Consequently, the displayed set number is not necessarily the only one with this problem Refer to the output file for the complete list of the sets with undefined GXZ. Notice that the warning message is only displayed when the program considers the material set to be orthotropic. A material set is considered orthotropic when:

One of the three modulus of elasticity (EX, EY and EZ) differs from the others (if EY or EZ are not defined, then the undefined modulus is considered to be equal to EX).

One of the three Poisson's ratios NUXY, NUYZ and NUXZ differs from the others (if NUYZ or NUXZ are not defined, then the undefined ratio is considered to be equal to NUXY).

One of the three coefficients of the thermal expansion (ALPHAX, ALPHAY and ALPHAZ) differs from the others (if ALPHAY or ALPHAZ are not defined, then the undefined coefficient is considered to be equal to ALPHAX)

Warning: Undefined NUXZ Is Set To 0 For Orthotropic Material Set: <Number>

STAR.EXE module.

This is only a warning message. For the specified orthotropic material set the undefined Poisson's ratio NUXZ is considered to be zero. This warning message is displayed for each set with this problem. Consequently, the displayed set number is not necessarily the only one with this problem.

*Refer to the output file for the complete list of sets with undefined NUXZ. For conditions which constitute a material set to be orthotropic, refer to the message: "**Warning: Undefined GXZ Is Equated To GXY For material set: <Number>**".*

WARRING - Zero Mass (Less Than <Value>) For Element <Number>

STAR.EXE Module

This is only a warning message indicating that the mass of the specified element is too small.

The mass of an element is either calculated from the density (material property) or as a point element, like MASS, is specified by the associated real constant set.

Warping Correction Applied For Small Warping At Element <Number>

STAR.EXE Module

This is only a warning message indicating that Warping Correction was applied in order to get acceptable results for this element. See also the comments for the error message. “Large warping, use QUAD4 element; twisted element<number>”

No action is needed.

Wrong Edge Of Element Has Zero Length, Element <Number> (Edge Connecting The Element <Node_order> And <Node_order > Nodes)

STAR.EXE Module

The edge specified by its two end nodes for the specified element has a length smaller than 1.e-30.

List the specified element and check the validity of its nodes and their connectivity. Check the coordinates of the nodes at the two ends of the specified edge. If the element is essentially a 4-noded element, then rearrange the nodes to form a triangular element with the third and fourth nodes collapsed. The ECHECK command (Meshing, Elements, Check Element) will fix this error.

Wrong Nodal Connectivity For Collapsed Element <Number> (Only Third And Forth Nodes May Be Collapsed. Use ECHECK)

STAR.EXE Module

The data for the specified element indicates that a triangular element is constructed from an originally rectangular element by collapsing two of its nodes.

Check whether the collapsed nodes are the third and forth nodes. Any other combination of collapsed nodes is prohibited. As is suggested in the error message you may simply use the ECHECK command to fix the problem.

Wrong Nodes Collapsed, At Element <Number> (Issue The ECHECK Command Prior To Running The Problem)

STAR.EXE Module

For the specified 4-noded element, wrong corner nodes are collapsed to form a triangular-shaped element. For this type of element, only the third and forth corner nodes are allowed to collapse. Refer to Element Library chapter in the COSMOS/M User’s Guide for details.

List the element and check the validity of its nodes and their connectivity. The ECHECK command (Meshing, Elements, Check Element) may fix this error.




Wrong Nodes Collapsed To Form Prism In Solid Element <Number> (Use The ECHECK Command To Fix The Problem)

STAR.EXE Module

The data for the specified element indicates that a prism is intended to be constructed from wrong nodes. Only nodes 3 and 8 may be collapsed on face 1 and nodes 7 and 8 on face 2, to form a prism. Any other combination of nodes is invalid.

List the element and check its nodal connectivity. Use the ECHECK command to fix the order of collapsed nodes and rerun the problem

Wrong Nodes Collapsed To Form Pyramid In Solid Element <Number>

STAR.EXE Module

The data for the specified element indicates that a pyramid is intended to be constructed from wrong nodes. Only nodes 5, 6, 7 and 8 may be collapsed to form a pyramid. Any other combination of nodes is invalid.

List the element and check its nodal connectivity. The ECHECK command may fix the problem

Zero Length Between Nodes 1-2 In Element <Number>

STAR.EXE and STRES.EXE Module

The length of the edge defined by the first and second corner nodes of the specified element is too small.

List the element and check whether the first and second nodes are coincident. If the model is actually too small, use a smaller unit of length. Note that the first and second nodes are not allowed to collapse for rectangular elements (only the third and fourth nodes may collapse).

Zero Length For Beam Element Number <Number> Execution Aborted

STAR.EXE or STRES.EXE Module

The length of the specified beam element is smaller than the allowable limit of 1.0E-20.

Make sure your dimensions are correct. If so, use smaller units and modify your material properties accordingly.




E References

References

1. Bathe, K-J. (1982). Finite Element Procedures in Engineering Analysis, Prentice-Hall Inc., Englewood Cliffs, NJ 07632.

2. Gross, B. and Srawley, J. E. (1972). Engineering Fracture Mechanics, V. 4, pp. 587-589.

3. Hertz, H. (1895). Gesammelte Werke, Vol. I, Leipzig.

4. Holland, R., and EerNisse, E. P. (1969). Design of Resonant Piezoelectric Devices, Research Monograph No. 56, The MIT Press, Cambridge, Massachusetts.

5. Irwin, G. R. (1961). “Analysis of Stresses and Strains Near the End of a Crack Traversing a Plate”, ASME Journal of Applied Mechanics, V. 29, pp. 651-654.

6. Keer, L. M., and Freedman, J. M. (1973). International Journal of Engineering Science, V. 11. pp. 1265-1275.

7. MacNeal, R. H., Editor. (1972). The NASTRAN Theoretical Manual, The MacNeal-Schwendler Corporation, Los Angeles, CA 90041.

8. Moon, F. C. (1984). Magneto-Solid Mechanics, John Wiley and Sons, New York.

9. Narayanaswami, R., and Adelman, H. (1977). “Evaluation of Tensor Polynomial and Hoffman Strength Theories for Composite Materials”, The Journal of Composite Materials, V. 2, p. 366.

10. Ugural, A. C., and Fenster, S. K. (1987). Advanced Strength and Applied Elasticity, Elsevier Science Publishing Company, New York.

11. Young, W. C. (1989). Roark's Formulas for Stress and Strain, Sixth Edition, McGraw Hill Book Company, New York.

12. ASME Boiler and Pressure Vessel Code, edition 1983, Section III, Division 1, subsection NB.

13. Kroenke, W. C., Addicott, G. W. and Hinton, B. M., “Interpretation of Finite Element Stresses According to ASME Section III” 75-PVP-63, *ASME Second National Congress on Pressure Vessels and Piping*, June 1975.

14. Kroenke, W. C., “Classification of Finite Element Stresses According to ASME Section III Stress Categories”, *Pressure Vessels and Piping, Analysis and Computer*, ASME, June 1974.

15. Gordon, J. L., “Outcur: An Automated Evaluation of Two-Dimensional Finite Element Stresses According to ASME Section III Stress Requirements,” Paper 76-WA/PVP-16, Dec. 1976.

16. ASME Boiler and Pressure Vessel Code, edition 1989, Section III, Division 1, Appendices.

Index

T

U

V

W

X