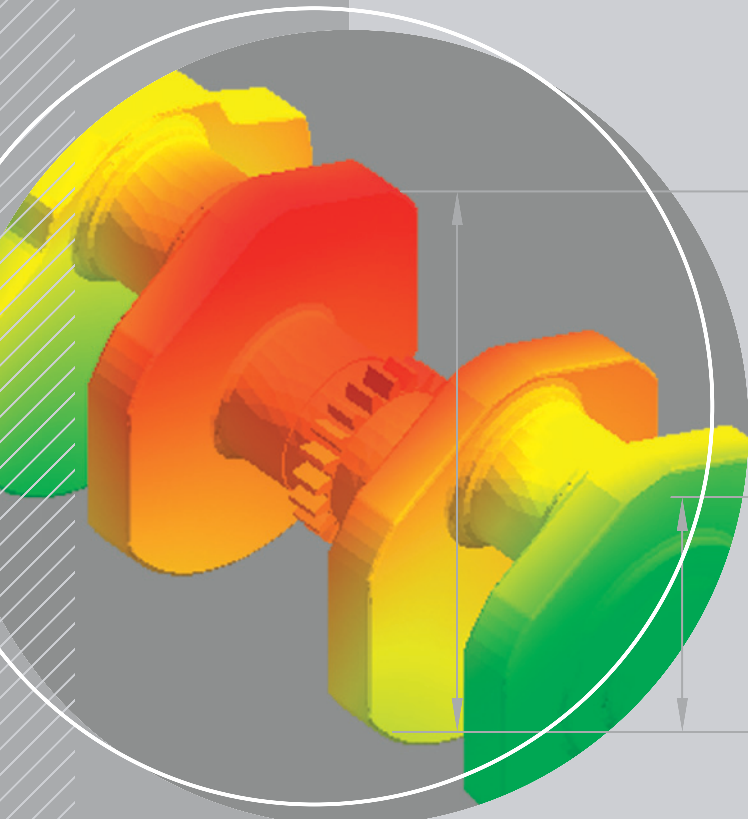
COSMOS™ 2004

DESIGN ANALYSIS MADE SIMPLE

Introducing COSMOSWorks



SolidWorks

© 2003 Structural Research and Analysis Corporation (SRAC)
12121 Wilshire Blvd., Suite 700
Los Angeles, California 90025-1170
310 207-2800 (phone)
310 207-2774 (fax)

All rights reserved.

Structural Research and Analysis Corp. (SRAC) is a Dassault Systemes S.A. (Nasdaq: DASTY) company.

Information is subject to change without notice. No material may be reproduced or transmitted in any form or by any means for any purpose without written permission of SRAC.

As a condition to your use of this software, you agree to accept the limited warranty, disclaimer and other terms and conditions set forth in SRAC License Agreement which accompanies this software. If, after reading the SRAC License Agreement, you do not agree with its terms and conditions, promptly return the unused software and all accompanying material to SRAC and your payment will be refunded.

COSMOS™, COSMOSWorks™, COSMOSMotion™, and COSMOSFloWorks™ are trademarks of SRAC.

ANSYS™ is a trademark of SAS IP.

MSC® and MSC/® are registered trademarks of MacNeal-Schwendler Corporation.

NASTRAN® is a registered trademark of the National Aeronautics and Space Administration.

PATRAN® is the registered trademark of PDA Engineering

Acrobat™, and Acrobat Reader™ are trademarks of Adobe Systems Incorporated.

IGES™ Access Library is a trademark of IGES Data Analysis, Inc. Other brand or product names are trademarks or registered trademarks of their respective holders.

Portions of this software © 2003 Solversoft, Inc.

Portions of this software © 2003 Simulog Technologies. A Business Unit of SIMULOG S. A.

Portions of this software © 2003 Computational Applications and System Integration, Inc.

Portions of this software © 2003 Unigraphics Solutions™, Inc.

Portions of this software © 2003 Visual Kinematics, Inc.

Portions of this software © 2003 DC MicroDevelopment, Inc.

Portions of this software © 1999, 2003 ComponentOne.

Portions of this software © 2003 D-Cubed Ltd.

Document Number: CWMISENG0703

Contents

Chapter 3 Design Studies

Chapter 4 Material Properties

Chapter 5 Loads and Restraints

Chapter 6 Meshing

Chapter 7 Design Optimization

Chapter 8 Viewing Results

Introduction

The COSMOSWorks Software

COSMOSWorks is a design analysis automation application fully integrated with SolidWorks. This software uses the Finite Element Method (FEM) to simulate the working conditions of your designs and predict their behavior. FEM requires the solution of large systems of equations. Powered by fast solvers, COSMOSWorks makes it possible for designers to quickly check the integrity of their designs and search for the optimum solution.

COSMOSWorks comes in several bundles to satisfy your analysis needs. It shortens time to market by testing your designs on the computer instead of expensive and time-consuming field tests.

This chapter discusses the following topics:

- **Intended Audience**
- **System Requirements**
- **Book Structure**
- **Conventions Used In This Book**

Intended Audience

The *Introducing COSMOSWorks* book is intended for new COSMOSWorks users. It assumes that you have basic SolidWorks skills.

This book introduces concepts and analysis processes in a high-level approach. It does not give step-by-step procedures on how to analyze models.

For step-by-step procedures, see the online help. For examples, click **Help, COSMOSWorks Online Tutorials**.

System Requirements

For the most recent information about system requirements, refer to the information provided in the *Read This First* sheet included in the box that contains the COSMOSWorks software CD.

Book Structure

The book is organized in chapters to reflect the main procedures in design analysis.

After introducing the basic concepts in design analysis in the *Fundamentals* chapter, the *Analysis Background* chapter introduces basic analysis concepts. Chapters following the Analysis Background chapter teach you basic skills of COSMOSWorks and how to use the software efficiently. The chapters are organized as follows:

Chapter	Title	Topics Discussed
1	Fundamentals	Introduces basic analysis concepts, COSMOSWorks terminology, and an overview of help options
2	Analysis Background	Provides background information on the various types of analyses available in COSMOSWorks
3	Design Studies	Introduces design study and design scenarios concepts, and explains study properties for different types of analyses.
4	Material Properties	Discusses topics related to assigning material properties to the model
5	Loads and Restraints	Options available for applying loads and restraints for different types of studies.
6	Meshing	Provides background information on meshing, mesh preferences, mesh control, contact options, and mesh failure diagnostics.
7	Design Optimization	Provides information on performing optimization studies.
8	Viewing Results	Outlines options and tools available for viewing results.

Conventions Used In This Book

This book uses the following conventions:

Convention	Meaning	Example
Bold Sans Serif	Any COSMOSWorks tool or menu item	Right-click **Mesh** and select **Create**.
Italic	References to books, chapters within the book, or to emphasize text.	Refer to the *Viewing Results* chapter for more details.
	Tip	 It is recommended to use **At Nodes** when using the p-method to solve static problems.

1

COSMOSWorks Fundamentals

This chapter presents information about the basic concepts and terminology used in **COSMOSWorks**. You will learn about the following topics:

- **What is COSMOSWorks?**. Introduces COSMOSWorks.
- **Benefits of Analysis**. Learn about the benefits of analysis.
- **Basic Concept of Analysis**. Learn about internal workings of COSMOSWorks.
- **Design Studies**. Lists the basic steps for performing analysis.
- **Material Properties**. Learn more about types of analysis COSMOSWorks offers.
- **Meshing**. Explore the concept of meshing and factors affecting it.
- **Running Studies**. Learn about the COSMOSWorks solvers and when to use them.
- **Parameters and Design Scenarios**. Learn about parameters and design scenarios.
- **Global and Local Coordinate Systems**. Describes how to express directional inputs and interpret directional output in global and local coordinate systems.
- **Design Check Wizard**. Describes how to check your design based on the analysis results.
- **Contact Problems**. Learn about this powerful function used in analyzing assemblies.
- **Working with Assemblies**. Lists tips for modeling assemblies.
- **Using Units**. Describes how to set default units and how they relate to SolidWorks units.

What is COSMOSWorks?

COSMOSWorks is a design analysis system fully integrated with SolidWorks. COSMOSWorks provides one screen solution for stress, frequency, buckling, thermal, and optimization analyses. Powered by fast solvers, COSMOSWorks enables you to solve large problems quickly using your personal computer. COSMOSWorks comes in several bundles to satisfy your analysis needs.

COSMOSWorks shortens time to market by saving time and effort in searching for the optimum.

What is SolidWorks?

SolidWorks™ is a mechanical design automation software that takes advantage of the familiar Microsoft Windows™ graphical user interface. This, easy-to-learn tool, makes it possible for you to quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

Benefits of Analysis

After building your design in SolidWorks, you need to make sure that it performs efficiently in the field. In the absence of analysis tools, this task can only be answered by performing expensive and time-consuming product development cycles. A product development cycle typically includes the following steps:

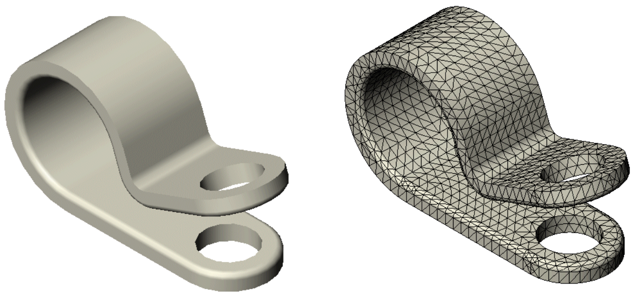
1. Build your model in the SolidWorks CAD system.
2. Prototype the design.
3. Test the prototype in the field.
4. Evaluate the results of the field tests.
5. Modify the design based on the field test results.

This process continues until a satisfactory solution is reached. Analysis can help you accomplish the following tasks:

- Reduce cost by simulating the testing of your model on the computer instead of expensive field tests.
- Reduce time to market by reducing the number of product development cycles.
- Improve products by quickly testing many concepts and scenarios before making a final decision, giving you more time to think of new designs.

Basic Concept of Analysis

COSMOSWorks uses the Finite Element Method (FEM). FEM is a numerical technique for analyzing engineering designs. FEM is accepted as the standard analysis method due to its generality and suitability for computer implementation. FEM divides the model into many small pieces of simple shapes called *elements* effectively replacing a complex problem by many simple problems that need to be solved simultaneously.



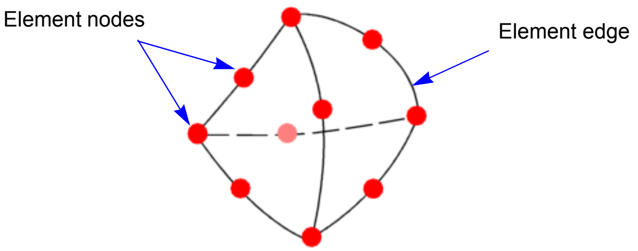
CAD model of a part

Model subdivided into small pieces (elements)

Elements share common points called nodes. The process of dividing the model into small pieces is called *meshing*.

The behavior of each element is well-known under all possible support and load scenarios. The finite element method uses elements with different shapes.

The response at any point in an element is interpolated from the response at the element nodes. Each node is fully described by a number of parameters depending on the analysis type and element used. For example, the temperature of a node fully describes its response in thermal analysis. For structural analyses using shells, the response of a node is described by three translations and three rotations. For structural analyses using tetrahedral elements, the response of a node is described by three translations. These are called degrees of freedom (DOFs). Analysis using FEM is called Finite Element Analysis (FEA).



A tetrahedral element. Red dots represent the element nodes. Edges of an element can be curved or straight

COSMOSWorks formulates the equations governing the behavior of each element taking into consideration its connectivity to other elements. These equations relate the response to known material properties, restraints, and loads.

Next, the program organizes the equations into a large set of simultaneous algebraic equations and solves for the unknowns.

In stress analysis, for example, the solver finds the displacements at each node and then the program calculates strains and finally stresses.

COSMOSWorks Professional offers the following types of studies:

Static Studies

Static studies calculate displacements, reaction forces, strains, stresses, and factor of safety distribution. Material fails at locations where stresses exceed a certain level. Factor of safety calculations are based on a failure criterion. COSMOSWorks offers four failure criteria.

Static studies can help you avoid failure due to high stresses. A factor of safety less than unity indicates material failure. Large factors of safety in a contiguous region indicate low stresses and that you can probably remove some material from this region.

Frequency Studies

A body disturbed from its rest position tends to vibrate at certain frequencies called natural, or resonant frequencies. The lowest natural frequency is called the fundamental frequency. For each natural frequency, the body takes a certain shape called mode shape. Frequency analysis calculates the natural frequencies and the associated mode shapes.

In theory, a body has an infinite number of modes. In FEA, there are theoretically as many modes as degrees of freedom (DOFs). In most cases, only a few modes are considered.

Excessive response occurs if a body is subjected to a dynamic load vibrating at one of its natural frequencies. This phenomenon is called resonance. For example, a car with an out-of-balance tire shakes violently at a certain speed due to resonance. The shaking decreases or disappears at other speeds. Another example is that a strong sound, like the voice of an opera singer, can cause a glass to break.

Frequency analysis can help you avoid failure due to excessive stresses caused by resonance. It also provides information to solve dynamic response problems.

Buckling Studies

Buckling refers to sudden large displacements due to axial loads. Slender structures subject to axial loads can fail due to buckling at load levels lower than those required to cause material failure. Buckling can occur in different modes under the effect of different load levels. In many cases, only the lowest buckling load is of interest.

Buckling studies can help you avoid failure due to buckling.

Thermal Studies

Thermal studies calculate temperatures, temperature gradients, and heat flow based on heat generation, conduction, convection, and radiation conditions. Thermal studies can help you avoid undesirable thermal conditions like overheating and melting.

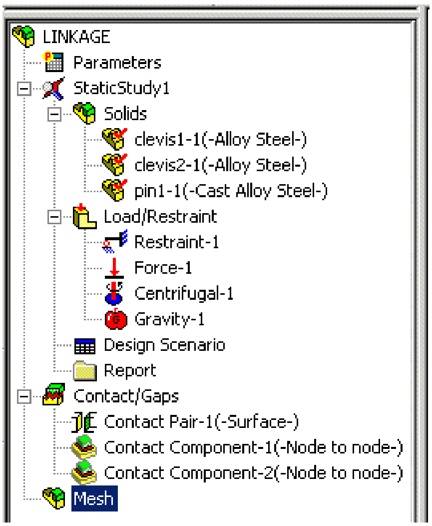
Optimization Studies

Optimization studies automate the search for the optimum design based on a geometric design. COSMOSWorks is equipped with a technology to quickly detect trends and identify the optimum solution using the least number of runs. Optimization studies require the definition of the following:

- **Objective**. State the objective of the study. For example, minimum material.
- **Design Variables or Geometry Constraints**. Select the dimensions that can change and set their ranges. For example, the diameter of a hole can vary from 0.5” to 1.0” while the extrusion of a sketch can vary from 2.0” to 3.0”.
- **Behavior Constraints**. Set the conditions that the optimum design must satisfy. For example, you can require that a stress component does not exceed a certain value and the natural frequency to be within a specified range.

COSMOSWorks Manager

The COSMOSWorks Manager tree organizes analysis studies. Its functionality is similar to the FeatureManager tree. You can use the menu system or the COSMOSWorks Manager tree to manage analysis studies. Because of its intuitive representation and context-sensitive right-mouse menus, the COSMOSWorks Manager is preferred over the menu system.



COSMOSWorks Manager tree

COSMOSWorks creates a folder in the COSMOSWorks Manager tree for each study. Sub folders define the parameters of the study. For example, each structural study has a **Load/ Restraint** subfolder. Each restraint and load condition is represented by an icon in this subfolder.

Right-mouse menus provide context-sensitive options. Drag and drop (or copy and paste) help you define studies quickly.

Restraints and loads use PropertyManager instead of dialog boxes, allowing your graphics to be displayed instead of hidden by dialog boxes.

Refer to the *Online Help* for more information about the COSMOSWorks Manager.

Design Studies

A model is usually subjected to different service environments and operational conditions during its life. It is therefore important to consider all possible scenarios of loads and boundary conditions and try different material properties in the analysis of a model. A study is defined by the following factors:

- model dimensions
- study type and related options to define analysis intent
- material properties
- loads and boundary conditions

To create a study, right-click the top icon in the COSMOSWorks Manager tree and click **Study**. Click **Add** to define a study by name, analysis type, mesh type, and properties. Mesh type is required for static, frequency, buckling, and thermal studies. The properties of the study set options related to a particular analysis type.

The mesh type sets the type of elements to be used for meshing. Although in theory you can use tetrahedral elements for all models, they are inefficient for thin models. Shell elements are naturally suitable for modeling thin parts.

Choosing the Mesh Type

Solid Mesh

Use the solid mesh for bulky models. All elements are tetrahedral with straight or curved edges.

Shell Mesh Using Mid-surfaces

Use this option for sheet metals and simple thin parts. The program extracts midsurfaces and assigns thickness automatically. You cannot specify more than one material for this option. Each element has a triangular shape with straight or curved edges and a uniform thickness.

Shell Mesh Using Surfaces

Use this option to mesh surface models or selected faces of parts and assemblies. You can assign a different thickness and material to each surface or face.

Using Design Studies

You can use design studies to check existing products or design new ones.

The COSMOSWorks line of products offers other types of studies like nonlinear, dynamic response, fluid flow, and electromagnetics.

Checking an existing design

When checking an existing product, the geometry is already determined. The goal is to check the performance of the product under different working conditions and investigate the possibility of improving the performance or saving material.

Making a new design

When using design analy4sis to make a new design, you can try different geometric configurations and materials to test the response of the model in various working conditions.

Analysis Steps

You complete a study by performing the following steps:

- Create a study defining its analysis type and options.
- If needed, define parameters of your study. Parameters could be a model dimension, a material property, a force value, or any other entity that you want to investigate its impact on the design.
- Define material properties. This step is not required in COSMOSWorks if material properties were defined in SolidWorks.
- Specify restraints. For example, in structural studies you define how the model is supported.
- Specify the loads.
- Mesh the model where COSMOSWorks divides the model into many small pieces called elements.
- Link the parameters to the appropriate study inputs.
- Define as many design scenarios as you want (up to 100 design scenarios).
- Run the study or selected design scenarios.
- View and list the results.

You can define material properties, loads, restraints, and create the mesh in any order. However, you must define all the necessary steps before running the study.

Optimization studies do not require meshing.

Material Properties

Before running a study, you must define all material properties required for the associated analysis type. For solid assemblies, each component can have a different material. For shell models defined with the *Shell using surfaces* option, each shell can have a different material and thickness.

There are four ways to define material properties:

- Use materials assigned to parts in SolidWorks,
- Pick a material from the COSMOS or SolidWorks Material Libraries,
- Specify the values of properties manually, or
- Pick a material from the Centor Material Library (an add-on option).

Refer to the *Material Properties* chapter for more details.

SolidWorks 2004 allows you to add materials and define visual and physical material properties in part documents. Physical properties are used by COSMOSWorks. Assigning a material to a part in COSMOSWorks does not update the material used in SolidWorks.

Material Editor

COSMOSWorks comes with a material editor. Use the material editor to add materials to the COSMOS Material Library or create your own libraries.

To learn how to use the Material Editor/Browser, refer to the *Material Properties* chapter.

Restraints and Loads

Restraints and loads define the environment of the model. Each restraint or load condition is represented by an icon in the COSMOSWorks Manager tree. COSMOSWorks provides context-sensitive options for defining restraints. For example, if all the selected faces are cylindrical or a reference axis is selected, the program expects you to define radial, circumferential, and axial restraints.

Loads and restraints are fully associative and automatically adjust to changes in geometry. The drag and drop (or copy and paste) functionality in the COSMOSWorks Manager tree lets you copy studies, folders, and items.

COSMOSWorks can import loads from COSMOSMotion and COSMOSFloWorks. Refer to the *Online help* or the *Loads and Restraints* chapter for details.

Dangled Restraints and Loads

If, after applying a restraints or load to an entity, you make geometry changes such that the entity is no longer defined, the restraint becomes dangled. COSMOSWorks gives you a message that lists dangled restraints or loads for each study you defined.

If rebuilding fails after a geometry change, all assignments (material, restraints, loads, mesh control, etc.) become invalid. However, you can still view results from an earlier analysis.

Connectors

A connector is a mechanism that defines how a face is connected to another face or to the ground. Connectors are encountered in many real life designs. Using connectors simplifies modeling. In many cases, you can simulate the desired behavior without having to create the detailed geometry or define contact conditions. Connectors contribute to stabilizing the model. You can define rigid, spring, pin, and elastic support connectors.

Meshing

Finite Element Analysis (FEA) provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes called *elements* connected at common points called *nodes*. The process of subdividing the model into small pieces is called *meshing*. Finite element analysis programs look at the model as a network of interconnected elements.

Meshing is a crucial step in design analysis. COSMOSWorks lets you create a mesh of solid elements (tetrahedral), or shells (triangular). The solid mesh is appropriate for bulky or complex 3D models. Shell elements are suitable for thin parts (like sheet metals).

The accuracy of the solution depends on the quality of the mesh. In general, the finer the mesh the better the accuracy. The generated mesh depends on the following factors:

- Type of mesh (solid, shell using midsurfaces, or shell using surfaces).
- Active mesh preferences.
- Mesh control.
- Contact conditions for static and thermal assembly problems.
- Global element size and mesh tolerance.

Mesh Preferences

Mesh preferences play an important role in meshing. It is recommended to check mesh preferences before meshing.

Mesh Control

Mesh control refers to using different element sizes at different regions of the model. COSMOSWorks provides mesh control at vertices, edges, faces, and components.

Contact Conditions

Contact conditions play an important role in meshing. Changes in contact conditions require remeshing.

Global Element Size

COSMOSWorks suggests a global element size and tolerance. The global element size refers to the average length of an element edge. The number of elements increases rapidly by using a smaller global element size.

Adaptive Methods for Static Studies

Adaptive methods help you obtain an accurate solution for static studies. COSMOSWorks offers two adaptive methods. The two adaptive methods in COSMOSWorks are the h-method, and the p-method.

The concept of the h-method is to use smaller elements in regions with high errors. After running the study and estimating errors, you can use mesh control to specify smaller element sizes in regions with high errors. You can continue this process until you are satisfied with the level of accuracy. To estimate the errors, plot the Error stress component for elements (not nodes). This method is not automated in this release.

Running Studies

After assigning materials, defining loads and restraints, and meshing the model, you can run the study.

To run a study, right-click the study folder icon in the COSMOSWorks Manager tree and select **Run** or click **Run** in the COSMOSWorks Main toolbar.

Solvers

COSMOSWorks offers different solvers to handle different types and sizes of problems more efficiently. The solvers exploit a new technology for the solution of large systems of simultaneous equations to reduce solution time, disk space, and memory requirements.

COSMOSWorks offers the following solvers:

- The **Direct Sparse** solver
- The **FFE** solver (iterative)
- The **FFEPlus** solver (iterative)

COSMOSWorks solvers are in many cases 100 times faster than conventional solvers. You select the solver when defining the properties of a study. In some cases, the program switches to another solver automatically if the selected solver does not support all of the options used in a study. All solvers should give similar answers provided that the same mesh is used. However, the performance and speed vary depending on the type and size of the problem. All solvers are efficient for small problems. The FFEPlus solver is particularly efficient for large problems (over 100,000 DOF).

Refer to the *Design Studies* chapter for more information.

Viewing Results

After running the analysis, COSMOSWorks generates standard plots for each type of analysis automatically. The standard plots for an analysis type are the most commonly used results. For example, after running a static study, COSMOSWorks creates result folders containing default plots for stress, strain, displacement, and deformation. You can view a plot by double-clicking its icon in the COSMOSWorks Manager tree.

You can also define other plots by right-clicking a result folder and selecting **Define**. When defining plots, you can use reference coordinate systems. For example, you can view radial and tangential stresses by selecting an axis when defining stress plots. You can associate result plots with named views.

COSMOSWorks result viewing tools include fringe plots, section plots, iso plots, animation, probing, and exploded views. For sections plots, you can choose planar, cylindrical, and/or spherical cutting tools. A clipping utility is provided for convenient viewing of section and iso plots.

For more information, refer to the *Viewing Results* chapter.

Generating Reports

You can generate a structured Internet-ready report that includes all available plots automatically. The report wizard guides you to customize the report and include result plots. To start the *Report* wizard, right-click the **Report** folder and select **Define**.

Saving Result Plots

You can save result plots in eDrawing, bitmap, VRML, XGL, and ZGL formats. You can save animations as AVI video files. You can include result plots automatically in the study report. To save a plot in any of these formats, right-click the plot icon and select **Save As**.

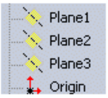
Parameters and Design Scenarios

You can define parameters and expressions for subsequent use in defining input. You define a parameter by a name, type, unit, and value or expression. A parameter can relate to geometry or analysis. Once you define a parameter of a certain type, you can use it to define input of the same type. For example, you can define **Force1** as a force parameter. When defining force values, you can link **Force1** to any force value field instead of entering a numeric value. Changing the value of **Force1**, automatically changes the applied force.

Design scenarios allow you to evaluate up to 100 what-if scenarios defined by model dimensions and analysis parameters. Due to the size of the disk space required to save all results for all scenarios, the program saves detailed results for the last design scenario and summary results for all other scenarios.

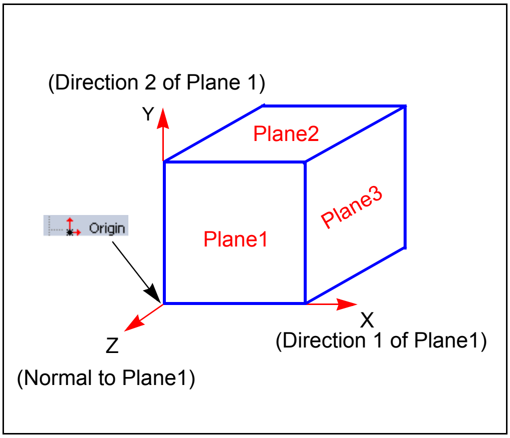
For more information about studies, refer to the *Design Studies* chapter.

Global and Local Coordinate Systems

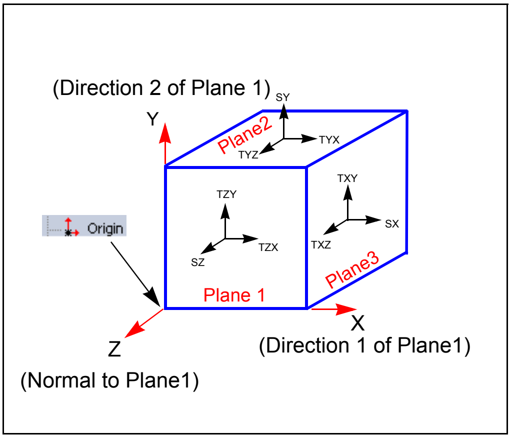


Directional input in COSMOSWorks refers by default to the global coordinate system (X, Y, and Z), which is based on Plane1 with its origin located at the Origin Origin of the part or assembly. Plane1 is the first plane that appears in the FeatureManager tree and can have a different name. The reference triad shows the global X-, Y-, and Z-directions.

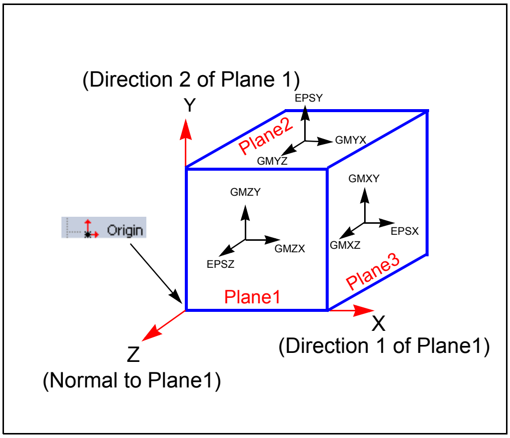
The figure illustrates the relationship between the global coordinate system and Plane1, Plane2, and Plane3.



Where X is Direction 1 of Plane1, Y is Direction 2 of Plane1, and Z is the Normal to Plane1. The two figures below illustrate stress and strain components in these directions.



Stress Components



Strain Components

Local coordinate systems are coordinate systems other than the global coordinate system. You can specify restraints and loads in any desired direction. For example, when defining a force on a cylindrical face, you can apply it in the radial, circumferential, or axial directions. Similarly if you choose a spherical face, you can choose the radial, longitude, or latitude directions. In addition, you can use reference planes and axes.

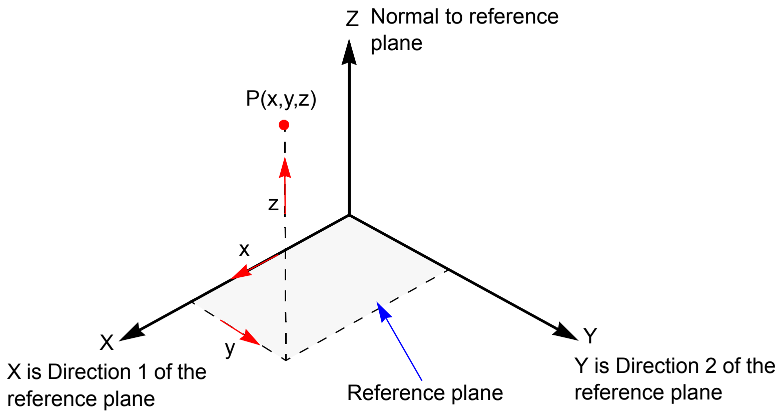
When viewing results, you can also use reference planes and axes. For example, you can view stresses on a cylindrical face in the radial direction.

Using Reference Planes and Axes

You can use reference planes and axes to define orthotropic material properties or apply directional loads and restraints.

Using Reference Planes

A reference plane defines a Cartesian coordinate system as illustrated in the figure.

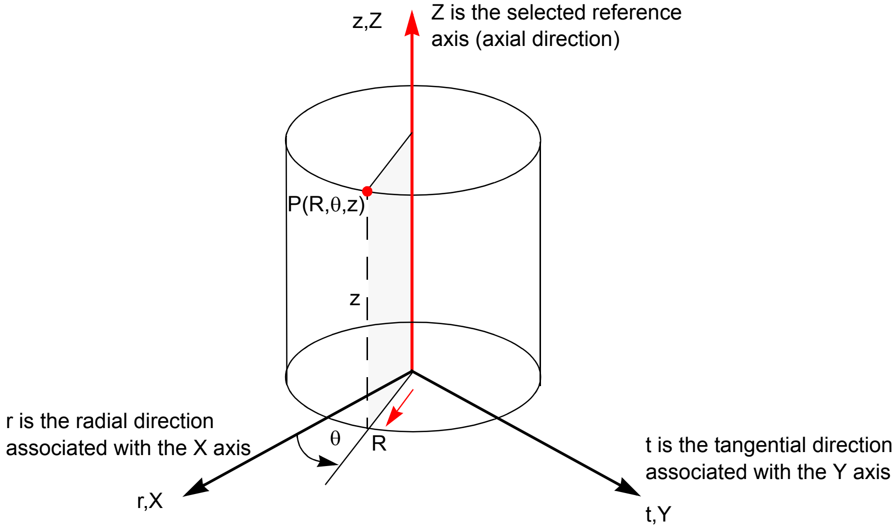


Cartesian Coordinate System

Using Reference Axes

A reference axis defines a cylindrical coordinate system as illustrated in the figure.

Refer to the *Loads and Restraints* chapter for more details.



Cylindrical Coordinate System

Design Check Wizard

For static studies, the **Design Check Wizard** guides you step-by-step to assess the safety of design based on a selected failure criterion. It calculates the factor of safety distribution throughout the model.

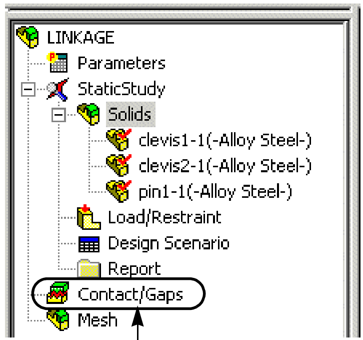
COSMOSWorks offers the following failure criteria:

- The *Maximum von Mises* Stress criterion
- The *Maximum Shear Stress* criterion
- The *Mohr-Coulomb Stress* criterion
- The *Maximum Normal Stress* criterion

Refer to the *Viewing Results* chapter for more details.

Contact Problems

COSMOSWorks supports contact conditions for static and thermal analyses of assembly models meshed with solid elements. You can consider the effect of friction between the contacting faces. A *Contact/Gaps* icon appears in the COSMOSWorks Manager tree.



A Contact/Gaps icon appears in assembly documents

Contact problems take a longer solution time than similar static problems without contact because contact iterations are needed to reach a solution.

By default, COSMOSWorks assumes that assembly components are bonded at their common regions. The user interface provides global, component, and local options to define contact conditions. Global settings apply where no component or local settings are defined. Component settings apply unless local settings are specified.

The local surface (face-to-face) contact condition allows you to simulate thermal contact resistance for thermal studies. A shrink fit contact condition is provided to simulate shrink fit problems.

A special option for large displacements is provided where the program uses a nonlinear approach to solve the problem.

Contact conditions are reflected on the mesh. A change in contact conditions requires remeshing the model.

For more information, refer to the *Meshing* chapter.

Result Databases

Analysis information are saved in database files. The database files for a study have a common name and different extensions. The database name for a study is constructed automatically by joining the study name to the part or assembly name separated by "-". For example if the document name is *Crank-assembly* and the study name is *Initial-Study*, then the database name for the study will be *Crank-assembly-Initial-Study.*

When running large models, the program can create large files. Result files are saved in the folder specified in the *Work directory* of the *Result* preferences dialog box. To change this folder, right-click the top icon in the COSMOSWorks Manager tree, select **Preferences**, and click the **Results** tab.

Result databases can occupy a large amount of disk space.

Working with Assemblies

When working with assemblies, note the following:

- Make sure that **automatic loading of components as lightweight** in **Tools**, **Options**, **System Options**, **Performance** is unchecked.
- While the automatic loading of assembly components as lightweight can improve performance of large assemblies significantly in modeling operations, it can cause serious errors when working with COSMOSWorks.
- Click **Tools**, **Interference Detection** to check interference.
- All parts should be free from interference with each other unless you plan to use the shrink fit contact option at the interfering boundaries.
- You can exclude components from analysis by suppressing them and then remeshing the model.
- Hiding components does not remove them from analysis.
- You can hide components during pre and postprocessing for improved viewing.
- You can create exploded views.

Using Units

COSMOSWorks allows you to choose the units for defining analysis data and viewing the results. You can set your preferred units by clicking **COSMOSWorks**, **Preferences**, and then clicking the **Units** tab. COSMOSWorks uses preferred units as the default units. COSMOSWorks displays the units it is using when defining the model or viewing the results.

The preferred units for COSMOSWorks and SolidWorks are independent of each other.

Setting the preferred system of units does not restrict you from using other units. In every step, COSMOSWorks allows you to use the appropriate desired units. For example, you can choose SI as your preferred system of units, specify pressure in psi, displacements in millimeters, view the displacement results in inches, and stress results in N/m^2.

The preferred units are not entirely defined by the system of units. You can set your preferred units for length, temperature, and angular acceleration independently. For example, you can select Metric, which uses centimeters as the unit of length, as your preferred system but choose inches and Fahrenheit as the length and temperature units, respectively. In this case, inch appears as the default unit in the *Restraint* dialog box, and Fahrenheit appears as the default unit for temperature input.

The *Meshing* PropertyManager displays the suggested average element size and the tolerance in the default unit of length in SolidWorks.

The *Export* tab in the *Preferences* dialog box allows you to specify your preferred system of units when exporting models to COSMOS.

2

Analysis Background

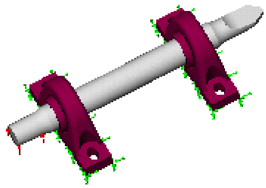
This chapter provides the basic theoretical information required for using COSMOSWorks. It explains what each type of analysis does, the underlying assumptions, the required input, and the expected output. It also gives a brief description of how to perform each type of analysis. The following topics are discussed:

- **Linear Static Analysis**
- **Frequency Analysis**
- **Linearized Buckling Analysis**
- **Thermal Analysis**
- **Optimization Studies**

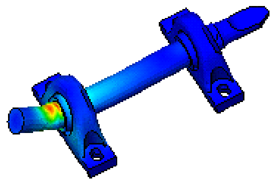
Linear Static Analysis

When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium.

Linear Static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.



Undeformed Model



Stress Plot on Deformed Model

Linear static analysis makes the following assumptions:

Static Assumption

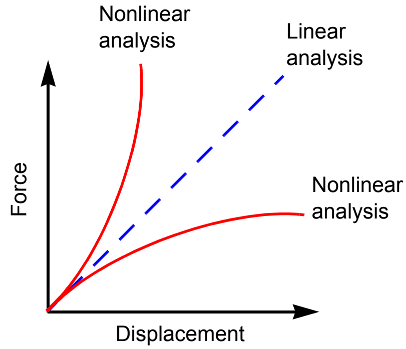
All loads are applied slowly and gradually until they reach their full magnitudes. After reaching their full magnitudes, loads remain constant (time-invariant). This assumption allows us to neglect inertial and damping forces due to negligibly small accelerations and velocities. Time-variant loads that induce considerable inertial and/or damping forces may warrant dynamic analysis. Dynamic loads change with time and in many cases induce considerable inertial and damping forces that cannot be neglected.

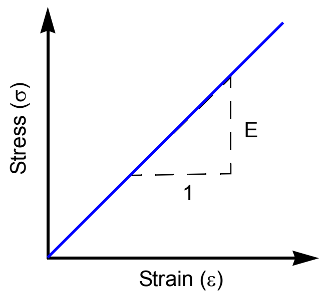
- It is important to verify the static assumption since a dynamic load can generate stresses of up to $1/(2\xi)$ times the stresses generated by static loads with the same magnitude, where ξ is the critical damping ratio. For a lightly damped structure with 5% damping, dynamic stresses are about 10 times larger than static stresses. The worst case scenario occurs at resonance. Refer to the section on *Frequency Analysis* in this chapter.
- You can use static analysis to calculate the structural response of bodies spinning at a constant velocity or travelling with a constant acceleration since the associated loads do not vary with time.
- You can use the *Dynamic Response* or the *Nonlinear Dynamic* analysis modules, available in other COSMOS products, to calculate the structural response due to dynamic loads. Dynamic loads include oscillatory loads, impacts, collisions, and random loads.

Linearity Assumption

The relationship between loads and induced responses is linear. For example, if you double the loads, the response of the model (displacements, strains, and stresses), will also double. You can make the linearity assumption if:

- all materials in the model comply with Hooke's Law, that is *Stress is directly proportional to Strain*.
- the induced displacements are small enough to ignore the change in stiffness caused by loading.
- boundary conditions do not vary during the application of loads. Loads must be constant in magnitude, direction, and distribution. They should not change while the model is deforming.

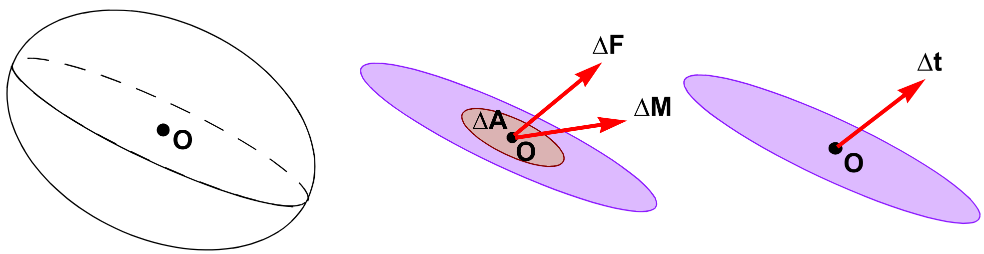




What is Stress?

The internal forces in a body vary from one point to the other. Across any small internal plane area, loads are exerted by the part of the body on one side of the area upon the part on the other side. Stress denotes the intensity of these internal forces (force per unit area).

Stress at a Point



Plane dividing the body into two parts

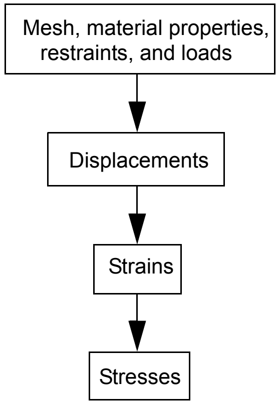
Resultant force and moment vectors on a region of area ΔA about O in plane

Limiting stress vector at point O in plane

In a continuous body, you can view the stress at a point as follows:

- Imagine an arbitrary plane that cuts through the body at that point,
- Consider an infinitesimally small area ΔA around that point on the plane,
- Suppose that the magnitude of the forces transmitted across ΔA in a certain direction is ΔF,
- The stress in that direction is then given by ΔF/ΔA as ΔA approaches 0.

Sequence of Calculations



Given a meshed model with a defined material properties, displacement restraints and loads, the linear static analysis module proceeds as follows:

- The program constructs and solves a system of linear simultaneous finite element equilibrium equations to calculate displacement components at each node.
- The program then uses the displacement results to calculate strain components.
- The program uses the strain results and the stress-strain relationships to compute stress components.

Stress Calculations

Stress results are first calculated at special points, called Gaussian points or Quadrature points, located inside each element. These points are selected to give optimal numerical results. The program calculates stresses at the nodes of each element by extrapolating the results available at the Gaussian points.

After a successful run, nodal stress results at each node of every element are available in the database. Nodes common to two or more elements have multiple results. In general, these results are not identical because the finite element method is an approximate method. For example, if a node is common to three elements, there can be three slightly different values for every stress component at that node.

When viewing stress results, you can ask for element stresses or nodal stresses. To calculate element stresses, the program averages the corresponding nodal stresses for each element. To calculate nodal stresses, the program averages the corresponding results from all elements sharing that node.

Required Input for Linear Static Analysis

To perform linear static analysis, you need the following:

- **Meshed model**. You must mesh the model before running the analysis. Contact conditions must be defined before meshing. Any change in geometry, contact conditions, or mesh options requires remeshing.

- **Material properties**. You must define the Young's Modulus (also called the Modulus of Elasticity). Poisson's Ratio is assumed to be zero if not defined. In addition, you need to define the density when considering the effect of gravity and/or centrifugal loading and the coefficient of thermal expansion when considering thermal loading. When you select a material from SolidWorks or COSMOS material libraries, these properties are assigned automatically. The default value used for the shear modulus is calculated from

$$GXY = \frac{EX}{2(1 + NUXY)}$$

  Compressive, tensile, and yield strength are used by failure criteria to assess failure. They are not used in calculating stresses. For orthotropic materials, you can define different Moduli of Elasticity, Shear Moduli, and Poisson Ratios and Coefficients of Thermal Expansion in different directions.

- **Restraints**. Adequate restraints or connectors (springs, elastic supports, pins, or rigid connectors) to prevent the rigid body motion of every part. If your model is not adequately constrained, check the **Use soft springs to stabilize the model** option in the **Static** dialog box. When importing loads from COSMOSMotion, check the **Use inertial relief** option. These options are available for the Direct Sparse and FFEPlus solvers.

- **Loads**. At least one of the following types of loading:

  Concentrated forces,

  Pressure,

  Prescribed nonzero displacements,

  Body forces (gravitational and/or centrifugal),

  Thermal (define temperature profile or import it from a thermal study or from COSMOSFloWorks),

  Imported loads from COSMOSMotion, or

  Import pressure from COSMOSFloWorks

When you create a study, click **Properties** in the **Study** dialog box to set the desired options. To modify the properties of an existing study, right-click on it in the COSMOSWorks Manager tree and click **Properties**.

Output of Static Analysis

By default, directions X, Y, and Z refer to the global coordinate system. If you choose a reference geometry, these directions refer to the selected reference entity.

- Displacement components:

UX	= Displacement in the X-direction
UY	= Displacement in the Y-direction
UZ	= Displacement in the Z-direction
URES	= Resultant displacement
RFX	= Reaction force in the X-direction
RFY	= Reaction force in the Y-direction
RFZ	= Reaction force in the Z-axis
RFRES	= Resultant reaction force

- Strain components and strain energy:

EPSX	= Normal strain in the X-direction
EPSY	= Normal strain in the Y-direction
EPSZ	= Normal strain in the Z-direction
GMXY	= Shear strain in the Y direction in the plane normal to X
GMXZ	= Shear strain in the Z direction in the plane normal to X
GMYZ	= Shear strain in the Z direction in the plane normal to Y
ESTRN	= Equivalent strain
SEDENS	= Strain energy density
ENERGY	= Total strain energy
E1	= Normal strain in the first principal direction
E2	= Normal strain in the second principal direction
E3	= Normal strain in the third principal direction

- Elemental and nodal stress components and related quantities. The following options are available:

SX	= Normal stress in the X-direction
SY	= Normal stress in the Y-direction
SZ	= Normal stress in the Z-direction
TXY	= Shear stress in the Y-direction in the plane normal to X
TXZ	= Shear stress in the Z-direction in the plane normal to X
TYZ	= Shear stress in the Z-direction in the plane normal to Y

The following quantities do not use reference geometry:

P1	= Normal stress in the first principal direction (largest)
P2	= Normal stress in the second principal direction
P3	= Normal stress in the third principal direction (smallest)
VON	= von Mises stress
INT	= Stress intensity = P1 - P3
ERR	= Relative error in stresses (available for element stresses only)

Equivalent Strain

Equivalent strain (ESTRN) is define as:

$$\mathrm{ESTRN} = 2.0\sqrt{\frac{\varepsilon_1 + \varepsilon_2}{3.0}}$$

Where:

$$\varepsilon_1 = 0.5[(\mathrm{EPSX} - \mathrm{meanstrain})^2 + (\mathrm{EPSY} - \mathrm{meanstrain})^2 + (\mathrm{EPSZ} - \mathrm{meanstrain})^2]$$

$$\varepsilon_2 = 0.25[\mathrm{GMXY}^2 + \mathrm{GMXZ}^2 + \mathrm{GMYZ}^2]$$

$$\mathrm{meanstrain} = (\mathrm{EPSX} + \mathrm{EPSY} + \mathrm{EPSZ})/3$$

Principal Stresses

Stress components depend on the directions in which they are calculated. For certain coordinate axis rotations, shear stresses vanish. The remaining three normal stress components are called principal stresses. The directions associated with *principal stresses* are called the *principal directions*.

Von Mises or Equivalent Stresses

The von Mises or equivalent stress is a stress quantity calculated from stress components. While the von Mises stress at a node does not uniquely define the state of stress at that node, it provides adequate information to assess the safety of the design for many ductile materials.

Unlike stress components, the von Mises stress has no direction. It is fully defined by magnitude with stress units. The von Mises stress is used by failure criteria to assess failure of ductile materials.

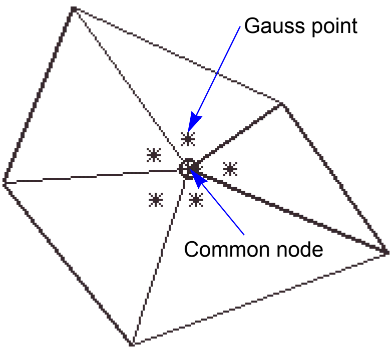
The von Mises stress is computed from the six stress components as follows:

$$\mathrm{VON} = \{(1/2)[(\mathrm{SX}-\mathrm{SY})^2 + (\mathrm{SX}-\mathrm{SZ})^2 + (\mathrm{SY}-\mathrm{SZ})^2] + 3(\mathrm{TXY}^2 + \mathrm{TXZ}^2 + \mathrm{TYZ}^2)\}^{(1/2)}$$

Or equivalently, from the three principal stresses,

$$\mathrm{VON} = \{(1/2)[(\mathrm{P1}-\mathrm{P2})^2 + (\mathrm{P1}-\mathrm{P3})^2 + (\mathrm{P2}-\mathrm{P3})^2]\}^{(1/2)}$$

Stress Error



For each element, stresses are calculated at locations called Gaussian or quadrature points. The results are then extrapolated to the nodes. Therefore, for a node common to several elements, each element will give stress results that are in general different than similar values from other elements. When you plot nodal stresses, the program averages these stress values to calculate the stress at the common node.

If the solution is exact, all elements give identical stress values at the common node. But, because FEA is an approximate method, the stresses will be different. The variation in stress values is used to estimate the error distribution throughout the model. If stresses from different elements at a node do not vary much, the error is low and if the variation is high, the error is high. COSMOSWorks makes these calculations for every node. Based on strain energy principles, COSMOSWorks estimates the errors in every element. Error estimation is based on the energy error norm and provides a valuable tool for estimating stress errors. The estimation is based on the variation in stress results at nodes common to two or more elements. The error decreases as the stress results at common nodes calculated from different elements approach each other. The stress error is available only if you select **Element values** under **Result type** in the **Stress Plot** dialog box.

For more information about error estimation, refer to *International Journal for Numerical Methods in Engineering*, vol. 24, 337-357 (1987) "*A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis*" by O.C. Zienkiewicz and J. Z. Zhu).

Thermal Stress Studies

Changes in temperature can induce substantial deformation, strains, and stresses. Thermal stress analysis refers to static analysis that includes the effect of temperature. COSMOSWorks lets you perform thermal stress analysis using one of the following options:

- Using a temperature profile specified directly by prescribing temperature values to faces, edges, and vertices in the static study.

- Using a uniform rise or drop in temperature for the whole model.
- Importing a temperature profile from a steady state or transient thermal analysis.
- Importing a temperature profile from COSMOSFloWorks.

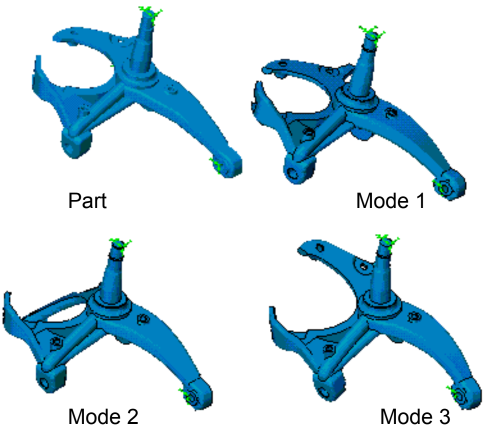
The **Include thermal effects** check box in the **Flow/Thermal Effects** tab in the **Static** dialog box controls the inclusion of thermal effects. If you define material properties manually, you must define the coefficient of thermal expansion for each material in the model.

Frequency Analysis

Every structure has the tendency to vibrate at certain frequencies, called natural or resonant frequencies. Each natural frequency is associated with a certain shape, called mode shape, that the model tends to assume when vibrating at that frequency. When a structure is properly excited by a dynamic load with a frequency that coincides with one of its natural frequencies, the structure undergoes large displacements and stresses. This phenomena is known as resonance. For undamped systems, resonance theoretically causes infinite motion. Damping, however, puts a limit on the response of the structures due to resonant loads.

A continuous model has an infinite number of natural frequencies. However, a finite element model has a finite number of natural frequencies that is equal to the number of degrees of freedom considered in the model.

The figure shows the lowest three modes of a model.



Example of Mode Shapes

The natural frequencies and corresponding mode shapes depend on the geometry, material properties, and support conditions. The effect of loads can be optionally considered. The computation of natural frequencies and mode shapes is known as modal, frequency, and normal mode analysis.

Effect of Loads on Frequency Analysis

When building the geometry of a model, you usually create it based on the original (undeformed) shape of the model. Some loads, like the structure's own weight, are always present and can cause considerable effects on the shape of the structure and its modal properties. In many cases, this effect can be ignored because the induced deflections are small.

COSMOSWorks gives you an option to consider the effect of applied loads on the modal properties by activating the **Use inplane effect** check box available in the **Frequency** dialog box. When running a frequency study with the **Use inplane effect** option selected, the program runs a linear static analysis first to calculate the deformed shape and then runs frequency analysis.

Tensile forces increase the natural frequencies of a structure and compressive forces reduce them.

To include the effect of loading on the resonant frequencies, you must use the Direct Sparse solver.

Required Input for Frequency Analysis

To perform frequency analysis, you need the following:

- **Meshed model**. You must mesh the model before running the analysis. The **Node-to-Node** and **Surface** contact conditions are not supported.
- **Material properties**. Similar to static analysis.
- **Number of modes**. The FFE and FFEPlus default settings calculate 5 modes in addition to any rigid body modes (modes with zero frequency or infinite period) available in the model automatically. Therefore, you do not have to apply any restraints. The rigid body modes are not counted among the requested number of modes. For example, if you ask for five modes for a free-free or unsupported model, FFE and FFEPlus extract six rigid body modes and five flexible modes.

- If you use the Direct Sparse solver, you must apply *adequate* restraints to stabilize your model or else you should activate the **Use soft spring to stabilize the model** option.
- Loads are not required and their effect is ignored unless you activate the **Use inplane effect** option.

Output of Frequency Analysis

When plotting displacements (mode shapes), directions X, Y, and Z refer to the global coordinate system. If you choose a reference geometry, these directions refer to the selected reference entity.

To list all requested resonant frequencies, click **COSMOSWorks, List Results, Mode Shape**.

Two folders are created in the COSMOSWorks Manager tree after a successful frequency analysis. These folders are:

- The **Displacement** folder

  You can plot displacement components on deformed or undeformed shapes:

  UX = Displacement in the X-direction

  UY = Displacement in the Y-direction

  UZ = Displacement in the Z-direction

  URES = Resultant displacement (does not use the reference geometry)

- The **Deformation** folder

  You can plot mode shapes. The corresponding frequency of the mode shape is shown on the plot.

Mode shapes illustrate the profile of the mode only (i.e., the displacement of nodes relative to each other). The displacement values are calculated based on various normalization procedures. COSMOSWorks normalizes each mode shape such that $\{\phi_i\}^T [M] \{\phi_i\}$ is equal to [I]. Where $\{\phi_i\}$ is the vector representing the i^{th} mode shape, $\{\phi_i\}^T$ is its transpose, [M] is the mass matrix, and [I] is the unit matrix.

Response to Dynamic Loads

Frequency analysis calculates the resonant frequencies and corresponding mode shapes only. The dynamic response module uses this information to calculate the dynamic response of your structure to loads.

Another option to calculate the dynamic response is to use the *Nonlinear Analysis* module. This module solves the dynamic response problem in the time domain and does not require the calculation of mode shapes and frequencies.

The dynamic response and nonlinear modules are available in advanced configurations of COSMOSWorks.

The effect of time-varying loads with frequencies less than 1/3 of the lowest resonant frequency of the model, can be approximated by static analysis in most cases.

Linearized Buckling Analysis

Models with thin parts tend to buckle under axial loading. Buckling can be defined as the sudden deformation which occurs when the stored membrane (axial) energy is converted into bending energy with no change in the externally applied loads. Mathematically, when buckling occurs, the total stiffness matrix becomes singular. The *Linearized Buckling* approach, used here, solves an eignvalue problem to estimate the c*ritical buckling factors* and the associated buckling shapes.

Buckling analysis calculates the smallest (critical) loading required to buckle a model. Buckling loads are associated with buckling modes. Designers are usually interested in the lowest mode because it is associated with the lowest critical load. When buckling is the critical design factor, calculating multiple buckling modes helps in locating the weak areas of the model. This may prevent the occurrence of lower buckling modes by simple modifications.

A more vigorous approach to study the behavior of models at and beyond buckling requires the use of nonlinear design analysis codes.

When to Use Buckling Analysis

Slender parts and assemblies with slender parts that are loaded in the axial direction buckle under relatively small axial loads. Such structures can fail due to buckling while the stresses are far below critical levels. For such structures, the buckling load becomes a critical design factor.

Buckling analysis is usually not required for bulky structures.

Required Input for Linearized Buckling Analysis

To perform linearized buckling analysis, you need the following:

- **Meshed model**. You must mesh the model before running the analysis. The **Node-to-Node** and **Surface** contact conditions are not observed by the solver.
- **Material properties, adequate restraints, and loads**. Similar to static analysis.

When you create a buckling study, click **Properties** in the **Study** dialog box to set the desired number of modes. FFEPlus and the Direct Sparse solver are available for buckling analysis. FFEPlus can calculate one buckling mode only. To modify the properties of an existing buckling study, right-click on it in the COSMOSWorks Manager tree and choose **Properties**.

Output of Linearized Buckling Analysis

When plotting displacements (mode shapes), directions X, Y, and Z refer to the global coordinate system. If you choose a reference geometry, these directions refer to the selected reference entity.

To list all requested buckling load factors, click **COSMOSWorks, List Results, Mode Shape**.

Two folders are created in the COSMOSWorks Manager tree after a successful buckling analysis run. These folders are:

- The **Displacement** folder, where you can plot buckling mode shape components on deformed or undeformed shapes:

  UX = Displacement in the X-direction

  UY = Displacement in the Y-direction

  UZ = Displacement in the Z-direction

  URES = Resultant displacement (does not use the reference geometry)

- The **Deformation** folder, where you can plot deformed mode shapes (without contours). The corresponding critical load factor of the plotted buckling mode will be displayed on the plot.

Mode shapes illustrate the profile of the mode only (i.e., the displacement of nodes relative to each other). The displacement values are calculated based on various normalization procedures. COSMOSWorks normalizes each mode shape such that $\{\phi_i\}^T [K_G] \{\phi_i\}$ is equal to [I], where $\{\phi_i\}$ is the vector representing the i^{th} mode shape, $\{\phi_i\}^T$ is its transpose, $[K_G]$ is the geometric stiffness matrix, and [I] is the unit matrix.

How to Interpret Results of Buckling Analysis

The critical load factor for a mode is the factor of safety against buckling in that mode. If the calculated critical load factor is greater than unity, the analysis indicates that the model will not buckle under the specified loads. The smallest loading under which the model will buckle can be calculated by multiplying all specified loads by the critical load factor.

For example, suppose that you applied the following loads:

- a force of 500 lbs on face 1, and
- a pressure of 250 psi on faces 2 and 3

If COSMOSWorks listed a critical load factor (factor of safety) of 2.3 for *mode 1***,** then, *assuming linear behavior*, buckling in *mode 1* occurs if you apply the following loads:

- a force of 500 x 2.3 = 1150 lbs on face 1

- a pressure of 250 x 2.3 = 575 psi on face 2 and face 3

If the critical load factor is negative, then buckling occurs only if you apply all loads in the opposite directions.

Thermal Analysis

Thermal analysis studies the flow of heat energy in a body.

Mechanisms of Heat Transfer

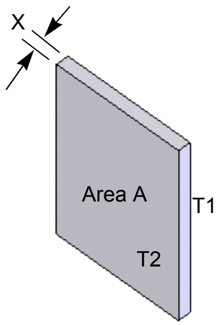
There are three mechanisms of heat transfer. These mechanisms are:

- Conduction
- Convection
- Radiation

Thermal analysis calculates the temperature distribution in a body due to some or all of these mechanisms. In all three mechanisms, heat energy flows from the medium with higher temperature to the medium with lower temperature. Heat transfer by conduction and convection requires the presence of an intervening medium while heat transfer by radiation does not.

Conduction

Conduction is the heat transfer mechanism in which thermal energy transfers from one point to another through the interaction between the atoms or molecules of the matter. Conduction occurs in solids, liquids, and gasses.



For example, a hot cup of coffee on your desk eventually cools down to the room temperature partly due to conduction from the coffee directly to the air and through the body of the cup.

Conduction does not involve any bulk motion of matter. The rate of heat conduction through a plane layer of thickness X is proportional to the heat transfer area and the temperature gradient, and inversely proportional to the thickness of the layer.

$$\dot{Q}_{\mathrm{Conduction}} = kA\frac{(T_1 - T_2)}{X} = kA\frac{dT}{dx}$$

where k, called the thermal conductivity, measures the ability of a material to conduct heat and dT/dx is the temperature gradient. The units of k are **W/m^{o}C** or **(Btu/s)/in^{o}F**.

Convection

Convection is the heat transfer mechanism by which heat energy transfers between a solid face and an adjacent moving fluid (or gas). Convection involves the combined effects of conduction and the moving fluid. The fluid particles act as carriers of thermal energy.

The rate of heat exchange between a fluid of temperature T_f and a face of a solid of area A and temperature T_s is expressed as:

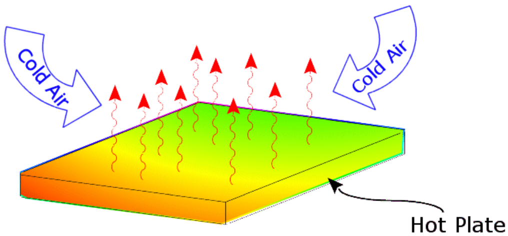
$$\dot{Q}_{Convection} = hA(T_s - T_f)$$

where h is the convection heat transfer coefficient, T_f is the temperature of the fluid away from the face of the solid. The units of h are **W/m^2.^{o}C** or **Btu/s.in^2.^{o}F**

Convection can be free or forced.

Free (Natural) Convection

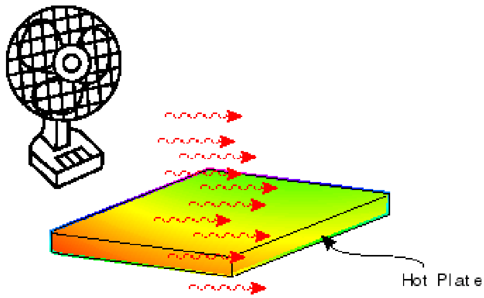
The motion of the fluid adjacent to a solid face is caused by buoyancy forces induced by changes in the density of the fluid due to differences in temperature between the solid and the fluid. When a hot plate is left to cool down in the air the particles of air adjacent to the face of the plate get warmer, their density decreases, and hence they move upward.



Natural Convection

Forced Convection

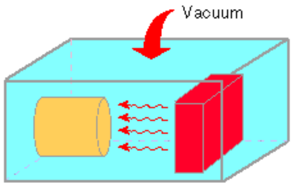
An external means such as a fan or a pump is used to accelerate the flow of the fluid over the face of the solid. The rapid motion of the fluid particles over the face of the solid maximizes the temperature gradient and results in increasing the rate of heat exchange.



Forced Convection

Radiation

Radiation is the thermal energy emitted by bodies in the form of electromagnetic waves. All bodies with temperatures above the absolute zero emit thermal energy. Because electromagnetic waves travel in vacuum, no medium is necessary for heat transfer due to radiation. The thermal energy of the sun reaches the earth by radiation. Since electromagnetic waves travel at the speed of light, radiation is the fastest heat transfer mechanism.



Generally, heat transfer by radiation becomes significant only at high temperatures.

COSMOSWorks considers surface to ambient and surface to surface radiation.

Stefan-Boltzmann Law

Stefan-Boltzmann's law states that the maximum rate of radiation that can be emitted by a surface of area A at a temperature T_s with a surrounding temperature T_e is given by:

$$\dot{Q}_{max} = \sigma A(T_s^4 - T_e^4)$$

where σ is the *Stefan-Boltzmann* constant (**5.67 x 10^{-8} W/m^2.C^4** or

3.3063 x 10^{-15} Btu/s.in^2.F^4).

A surface that is emitting heat energy at this rate is called a blackbody.

The ratio of the power per unit area radiated by a surface to that radiated by a blackbody at the same temperature is called emissivity (ε). A blackbody therefore has an emissivity of 1 and a perfect reflector has an emissivity of 0.

$$\dot{Q} = f\varepsilon\sigma A(T_s^4 - T_e^4)$$

Where f is the view factor and ε is the emissivity.

You can specify a different emissivity for each face participating in the radiation process by applying radiation for each face separately. COSMOSWorks considers radiation for all radiating faces in spite of grouping when applying the condition.

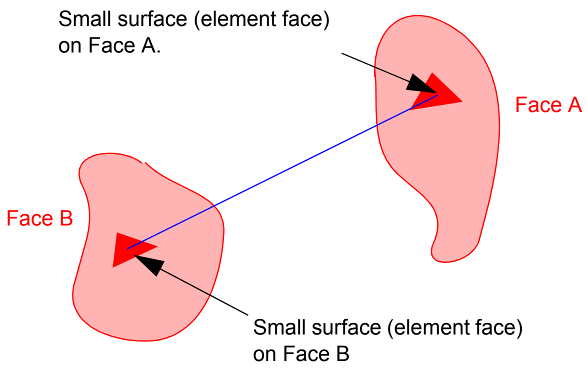
Surface- to-Surface Radiation

The rate of thermal energy exchange between two radiating faces depends on their temperatures, view factors, and emissivities.

Radiation View Factors

View factors, also known as shape or configuration factors, play a direct role in heat transfer due to radiation. The view factor F_{ij} between two small areas A_i and A_j is defined as the fraction of the radiation leaving area A_i that is intercepted by area A_j. In other words, F_{ij} represents how well area A_i sees area A_j. The view factor F_{ij} depends on the orientation of the small areas A_i and A_j and the distance between them. In general, F_{ij} is not equal to F_{ji}. A view factor ranges from 0 (no interception) to 1.0 (full interception).

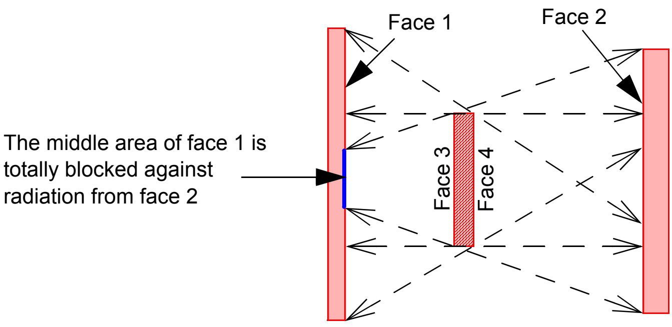
While many heat transfer books provide view factors for simple-shaped areas, calculating view factors for real life problems requires major effort even for fast computers. When requested to calculate view factors for the radiation between two faces A and face B, COSMOSWorks thinks of each face as composed of small areas (surfaces) defined by element faces. It then calculates a view factor for each element face relative to every other element face in the set. View factors between element faces on the same geometrical face are included in these calculations.



A concave face with a reasonably fine mesh can radiate to itself. Planar and convex faces do not radiate to themselves. Such effects are automatically considered.

Blocking

The radiation between two element faces can be blocked by a third element face. In that case, the view factor becomes 0. COSMOSWorks automatically considers blocking among all specified faces as illustrated in the figure. For proper blocking considerations, you must select faces participating in the radiation. In the example below, faces 3 and 4 partially block the radiation between faces 1 and 2. Incorrect results are obtained if you select faces 1, 2, and 3 only.



Considering blocking in radiation by participating faces

Open Versus Closed Radiation Systems

An open radiation system considers radiation heat exchange with the ambient in addition to surface-to-surface radiation. A closed radiation system considers surface-to-surface radiation only and ignores radiation heat exchange with the ambient.

Types of Heat Transfer Analysis

There are two types of heat transfer:

Steady State Thermal Analysis

In this type of analysis, we are only interested in the thermal conditions of the body when it reaches a state of thermal equilibrium. We are NOT interested in knowing the time it takes to reach this state. At thermal equilibrium, the thermal energy entering each point in the model is equal to the thermal energy leaving it. Generally, the only material property that is needed for steady state analysis is the thermal conductivity.

Transient Thermal Analysis

In this type of analysis, we are interested in knowing the thermal status of the model as a function of time. A thermos designer, for example, knows that the temperature of the fluid inside will eventually be equal to the room temperature (steady state), but he or she is interested in finding out the temperature of the fluid as a function of time. In specifying material properties for transient thermal analysis, you need to specify thermal conductivity, density, and specific heat. In addition, you need to specify the initial temperatures, solution time, and time increment.

For transient studies, you can associate heat flux and heat power conditions with a thermostat defined by a temperature range at a specified location. For each time step, the program turns the heat power or heat flux condition on or off based on the temperature at the thermostat location.

Required Input for Thermal Analysis

To perform thermal analysis, you need the following:

- **Meshed model**. You must mesh the model before running the analysis. The **Node-to-Node** contact condition results in isolated contact faces. The **Surface** contact condition supports thermal resistance.
- **Material properties**. You must define the thermal conductivity for steady state thermal studies. Density and specific heat may also be required for transient studies.
- **Loads and boundary conditions**. You can prescribe temperature at faces, edges, or vertices. Specify thermal energy as heat flux or heat power. Convection and radiation are applied as boundary conditions. When specifying convection, you need to enter the convection coefficient and the ambient temperature of the fluid or gas. Similarly, for radiation, you need to specify the emissivity and the surrounding temperature. The *Stefan-Boltzmann* constant is automatically defined by COSMOSWorks.
- **Solution parameters**. When creating a thermal study, you must specify whether you want to run steady state or transient analysis. Also, you may choose The motion of the fluid adjacent to a solid face is caused by buoyancy forces induced
- by changes in the density of the fluid due to differences in temperature between
- the solid and the fluid. When a hot plate is left to cool down in the air the particles
- of air adjacent to the face of the plate get warmer, their density decreases, and
- hence they move upward.FFE, FFEPlus, or Direct Sparse solver.

For transient studies, you need to specify the **Total time** of the analysis, the **Time increment**, and the **Initial temperature**. To define a thermostat, you need to specify a temperature range and the location of the thermostat.

Output of Thermal Analysis

By default, directions X, Y, and Z refer to the global coordinate system. If you choose a reference geometry, these directions refer to the selected reference entity.

A folder is created in the COSMOSWorks Manager tree after a successful thermal analysis run. This folder lets you plot the temperatures, temperature gradients, and heat flux.

Thermal Results

TEMP	= Temperature
GRADX	= Temperature gradient in the X-direction
GRADY	= Temperature gradient in the Y-direction
GRADZ	= Temperature gradient in the Z-direction
GRADN	= Resultant temperature gradient

HFLUXX	= Heat flux in the X-direction
HFLUXY	= Heat flux in the Y-direction
HFLUXZ	= Heat flux in the Z-direction
HFLUXN	= Resultant heat flux

where:

$$\mathrm{GRADN} = \sqrt{\mathrm{GRADX}^2 + \mathrm{GRADY}^2 + \mathrm{GRADZ}^2}$$

$$\mathrm{HFLUXN} = \sqrt{\mathrm{HFLUXX}^2 + \mathrm{HFLUXY}^2 + \mathrm{HFLUXZ}^2}$$

Optimization Studies

Optimization studies help you automate the search for the optimum solution. In optimizing a design, you need to define your objective (objective function), the dimensions of the design that can change (design variables), and the conditions that the design must satisfy (behavior constraints).

For example, you may want to vary some of the dimensions in your model to minimize the material, while maintaining a safe level of stresses. In this case, your objective is to reduce the volume of the material, the varying dimensions are the design variables, and the condition that the stress level cannot exceed a certain limit is the behavior constraint.

COSMOSWorks exploits the parametric, feature-based modeling, and the automatic regeneration capabilities of SolidWorks to automate the optimization process. It quickly detects the effects of changing design variables to minimize the number of design cycles leading to the optimum design.

For more information on optimization studies, refer to the *Design Optimization* chapter. For an example, refer to the *COSMOSWorks Online Tutorial*.

3

Design Studies

The concept of design studies lies at the heart of the operation of COSMOSWorks. In this chapter, you learn about the following topics:

- **Design Studies**. Understand what a design study is and learn the underlying concepts.
- **Study Types**. Explore different study types and what results each type offers.
- **Mesh Types**. Understand the available mesh types and how to choose the proper option.
- **Properties of Static Studies**. Learn about the options available for static studies.
- **Properties of Frequency Studies**. Learn about the options available for frequency studies.
- **Properties of Buckling Studies**. Learn about the options available for buckling studies.
- **Properties of Thermal Studies**. Learn about the options available for thermal studies.
- **Properties of Optimization Studies**. Learn about options available for optimization studies.
- **Parameters**. Overview the usage of parameters in analysis.
- **Design Scenarios**. Learn about multiple design scenarios for a study.
- **Multiple Studies**. Learn how to manage multiple studies.
- **Running Studies**. Learn how to run individual studies and multiple design scenarios.
- **Exporting Studies**. Learn how to export your COSMOSWorks model to other commercial FEA programs.

Design Studies

A model is usually subjected to different service environments and operational conditions during its life. It is therefore important to consider all possible scenarios of loads and boundary conditions and try different material properties in the analysis of a model.

A design study is defined by the following factors:

- model dimensions
- study type and related options to define the analysis intent
- material properties
- loads and boundary conditions

Study Types

Static (Stress) Studies

Static studies calculate displacements, reaction forces, strains, stresses, and factor of safety distribution. Material fails at locations where the stresses exceed a certain level. Factor of safety calculations are based on a failure criterion. COSMOSWorks offer four failure criteria.

Static studies can help you avoid failure due to high stresses. A factor of safety less than unity indicates material failure. Large factors of safety in a contiguous region indicates that you probably can remove material from this region.

Frequency Studies

Frequency studies calculate resonant frequencies and the associated mode shapes. When a body is subject to a vibrating environment, frequency studies can help you avoid failure due to excessive stresses caused by resonance

A body disturbed from its rest position tends to vibrate at certain frequencies called natural, or resonant frequencies. The lowest natural frequency is called the fundamental frequency. For each natural frequency, the body takes a certain shape called mode shape. Frequency analysis calculates the natural frequencies and the associated mode shapes.

In theory, a body has an infinite number of modes. In FEA, there are theoretically as many modes as degrees of freedom (DOFs). In most cases, only a few modes are considered.

Excessive response occurs if a body is subjected to a dynamic load vibrating at one of its natural frequencies. This phenomenon is called resonance. For example, a car with an out-of-balance tire shakes violently at a certain speed due to resonance. The shaking decreases or disappears at other speeds. Another example is that a strong sound, like the voice of an opera singer, can cause a glass to break.

Frequency analysis can help you avoid failure due to excessive stresses caused by resonance. It also provides information to solve dynamic response problems.

Buckling Studies

Buckling refers to sudden large displacements due to axial loads. Slender structures subject to axial loads can fail due to buckling at load levels lower than those required to cause material failure. Buckling can occur in different modes under the effect of different load levels. In many cases, only the lowest buckling load is of interest.

Buckling studies can help you avoid failure due to buckling.

Thermal Studies

Thermal studies calculate temperatures, temperature gradients, and heat flow based on heat generation, conduction, convection, and radiation conditions. Thermal studies can help you avoid undesirable thermal conditions like overheating and melting.

Optimization Studies

Optimization studies automate the search for the optimum design based on a geometric model. COSMOSWorks is equipped with a technology to quickly detect trends and identify the optimum solution using the least number of runs. Optimization analysis requires the following input:

- **Objective**. State your objective. For example, minimum material.
- **Design Variables or Geometry Constraints**. Select the dimensions that can change and set their ranges. For example, the diameter of a hole can vary from 0.5” to 1.0” while the extrusion of a sketch can vary from 2.0” to 3.0”.
- **Behavior Constraints**. Set the conditions that the optimum design must satisfy. For example, stresses, displacements, temperatures should not exceed certain values and the natural frequency should be in a specified range.

For more information refer to the *Design Optimization* chapter

Mesh Types

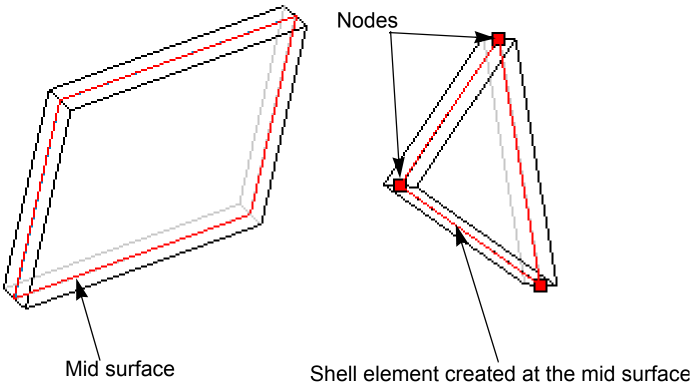
When creating a study, you select the mesh type. The mesh type sets the meshing procedure and the shape of elements to be used in meshing the model.The following options are available:

Solid

Use this option for bulky models. During meshing, COSMOSWorks creates tetrahedral elements. This option is not recommended for sheet metals and thin models because too many elements can be required to mesh the model accurately.

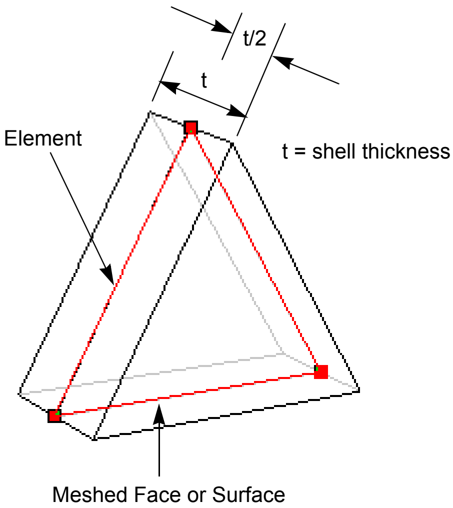
Shell mesh using mid-surfaces

Use this option for sheet metals and simple thin solid parts with one material. During meshing, COSMOSWorks creates shell elements based on midsurfaces. The thickness of elements is calculated automatically based on surface pairs. This option is not available for assemblies and surface models and can fail to generate the proper mesh for complex parts and parts with intersections. View the mesh and see if it represents the actual model before proceeding with the solution.



Shell mesh using surfaces

This option gives you full control on what faces or surfaces to mesh and what thickness and material to use for each face or surface. It is available for solid parts, assemblies, and surface models. Shell elements are placed such that the associated face or surface is located at the middle of the element across the thickness as illustrated in the figure.



For exact modeling, you need to modify the dimensions of the solid model to fit your analysis intent. For example, if you build a solid part of a 0.2” thick cylinder with an inner radius of 5”, you can mesh the inner or outer cylindrical faces. If you mesh the inner face and specify a shell thickness of 0.2”, you will be solving a cylinder with an inner radius of 4.9” and an outer radius of 5.1”. Similarly, if you mesh the outer face, you will be solving a cylinder with an inner radius of 5.1” and an outer radius of 5.3”.

Properties of Static Studies

The **Static** dialog box sets analysis properties for static studies. The dialog box has three tabs.

- The **Options** tab sets general static analysis options.
- The **Adaptive** tab sets parameters related to adaptive analysis.
- The **Flow/Thermal Effects** tab sets parameters related to importing temperatures or pressure loads from COSMOSFloWorks or temperatures from a thermal study.
- The **Remark** tab allows you to attach a remark to the study and include it in the study report.

Gap/Contact

Faces can be initially in contact or they can come into contact due to the effect of applied loads. The behavior of contacting faces is defined by the following factors:

- Global, component, and local contact conditions incorporated in the mesh of the model. Refer to the *Meshing* chapter for details about setting contact conditions.
- Options specified in the study properties.

The **Static** properties dialog box of static studies sets the following options related to contact:

Include Friction

Friction affects the behavior of contacting faces. Smooth contacting faces do not resist sliding against each other. Rough surfaces resist sliding. The coefficient of static friction determines the resistance to sliding. COSMOSWorks calculates friction force at a contact point by multiplying the normal contact force, induced by loading, by the specified coefficient of friction. The direction of the friction force at a location is opposite to the direction of the resultant displacement at that location.

Convergence of contact problems slows down as the coefficient of friction becomes larger.

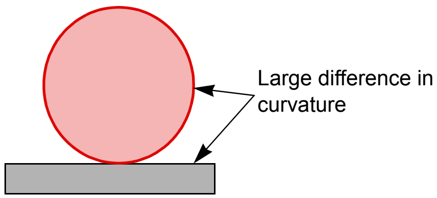
Ignore Clearance for Surface Contact

A gap between faces set for surface contact can exist in the initial configuration. The gap can be caused by modeling tolerance or by specifying local contact between faces.

This check box gives you the option to specify that faces set for surface contact behave as if a rigid body is inserted between them.

Option Unchecked:

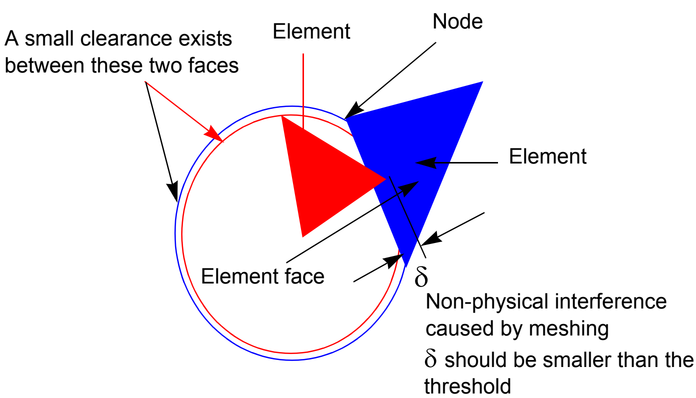
If the clearance between faces set for surface contact is less than 0.5% of the global element size, COSMOSWorks automatically assumes that the faces are initially in contact. Contact forces develop at points where the faces move closer to each other. If you specify a local contact between two faces that are initially more than 0.5% of the element size apart, contact forces develop only if the faces actually come into contact with each other. The flag should not be turned on when the two faces set for contact have large curvature differences.



Option Checked:

COSMOSWorks ignores the 0.5% of the element size threshold. There is no limitation on the initial gap between faces set for surface contact. The faces can move away from each other but they cannot get any closer to each other than in the initial configuration. The faces behave as if a rigid object is inserted between each source and target.

This flag should be unchecked (default) for most applications. It is designed to take care of special situations where meshing causes part interference, even though a clearance exists between the geometry of the parts as illustrated in the figure.



Turn on the flag in such cases

To minimize the possibility of this situation, use mesh control to specify a finer mesh in contact regions with this condition. High quality meshes (parabolic elements) are recommended for all contact problems.

This flag is global. You cannot turn it on for some surface contact pairs and off for others. For models containing contact pairs with large curvature differences as well as contact faces with tolerance, we recommend to uncheck the **Ignore clearance for surface contact** check box and use mesh control to specify a finer mesh in contact regions with tolerance.

Large Displacements

In surface contact, the program internally assigns one surface as a *source* and the other as *target*. For each node on the *source*, COSMOSWorks assigns one or more element faces on the *target*.

Contact is supported for solid elements only. It is not supported for shell elements.

This option determines how COSMOSWorks proceeds with the contact problem.

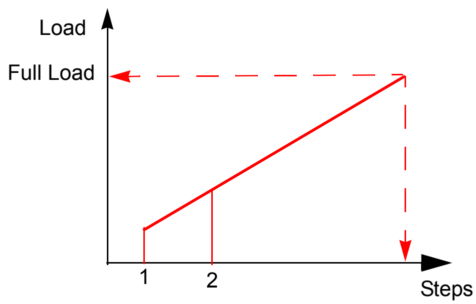
Option Unchecked:

The full load is applied at once. Source and target pairs are set based on the initial configuration and remain unchanged during contact iterations. Normals to contact areas are also based on the initial configuration and remain unchanged during contact.

This approach can lead to inaccurate results or convergence difficulties if these assumptions are not valid, a likely situation when displacements become large.

Option Checked:

Loads are applied gradually and uniformly in a number of steps up to their full values. The number of steps are decided internally by the program based on deformation results. Source and target pairs and normal to contact areas are evaluated at each solution step.



Recommended Procedure:

The following procedure is recommended for solving contact problems:

1 Solve the problem without activating the **Large displacement contact** flag.

2 Activate the **Large displacement contact** flag and try again in the following cases:

- If displacements or the orientations of the contacting faces are noticeable when the deformed shape is plotted using a scale factor of 1.0.
- If the highest strain exceeds 4%.
- If von Mises stresses are close to yield.

- Large displacement analysis is used with surface contact only. It does not support any other contact options.
- Full nonlinear analysis is required to handle large strain and material nonlinearity.

Flow/Thermal Effects

An unrestrained body expands or contracts freely as it is heated or cooled. The change in temperature causes strains but no stresses. If the body is prevented from free expansion or contraction, stresses are induced. The induced stresses are equivalent to those required to deform an equivalent unrestrained body similarly.

It is important to include the contribution of temperature changes to stresses for restrained models. The coefficient of thermal expansion material property is required to consider this effect. You need to specify the reference temperature associated with the stress-free condition. The following options are available in the **Flow/Thermal Effect**s tab of the Static dialog box:

Include Thermal Effects. Considers thermal loading for the analysis. Select one of the following thermal options:

Input Temperature. Considers the prescribed temperatures defined for the model.

When using this option, make sure to specify temperatures on components or shells. Specifying temperatures on the boundary only may not be practical since a temperature of zero is assumed at all other locations. If you define temperature on boundary only, you may need to create and solve a thermal study first to compute temperatures at all nodes.

Temperatures from thermal study. Reads the temperature values from a thermal study. Select a Thermal study and the Time step number (for a transient thermal study). The meshes of the thermal and static studies must be the identical.

Uniform temperature. Considers a uniform temperature at all nodes in the part. Specify the uniform temperature value.

Temperatures from COSMOSFloWorks. Reads the temperature values resulting from a completed COSMOSFloWorks on the same configuration from a file. Select the desired result file (*.fld) that has been generated by COSMOSFloWorks. SolidWorks model name, configuration name, and time step number associated with the specified file are displayed.

The **Reference temperature at zero strain** field sets the temperature at which no strains exist in the model.

Include fluid pressure effects from COSMOSFloWorks. Reads the pressure results from a COSMOSFloWorks result file.

Solvers

In finite element analysis, the problem is represented by a set of algebraic equations that must be solved simultaneously. There are two classes of solution methods: direct and iterative.

Direct methods solve the equations using exact numerical techniques. Iterative methods solve the equations using approximate techniques where in each iteration, a solution is assumed and the associated errors are evaluated. The iterations continue until the errors become acceptable.

COSMOSWorks offers the following choices:

- Direct Sparse solver
- FFE (iterative)
- FFEPlus (iterative)

FFE and FFEPlus use different equation reordering and data storage techniques to solve the problem.

Choosing a Solver

In general, all solvers give comparable results if the options are supported. While all solvers are efficient for small problems (25,000 DOFs or less), there can be big differences in performance (speed and memory usage) in solving large problems.

If a solver requires more memory than available on the computer, the solver uses disk space to store and retrieve temporary data. When this situation occurs, you get a message saying that the solution is going out of core and the solution progress slows down. If the amount of data to be written to the disk is very large, the solution progress can be extremely slow.

The following factors help you choose the proper solver:

- **Size of the problem**. In general, FFEPlus is faster in solving problems with over 100,000 DOF. It becomes more efficient as the problem gets larger.
- **Computer resources**. The direct sparse solver in particular becomes faster with more memory available on your computer.
- **Analysis options**. For example, the inplane effect, soft spring, and inertial relief options are not available if you choose the FFE solver.

- **Element type**. For example, contact problems and thick shell formulation are not supported by the FFE solver. In such cases, the program switches automatically to FFEPlus or the Direct Sparse solver.
- **Material properties**. When the moduli of elasticity of the materials used in a model are very different (like Steel and Nylon), iterative solvers are inherently less accurate than direct methods. The direct solver is recommended in such cases.

FFE Plus Convergence Parameters

FFEPlus is an iterative solver. It first assumes a solution for all unknowns and then improves it in each iteration until the solution converges. Convergence is controlled by two parameters: maximum number of trials and the stopping threshold (convergence tolerance).

MaxImum Number of Iterations

The solver stops if it does not achieve convergence within the specified maximum number of iterations.

Stopping Threshold

The solver stops if the solution does not converge within the allowed number of iterations. A smaller tolerance and a higher maximum number of iterations give you more accurate results but the program takes more time. User interference is not needed in most cases. However, you can reset the convergence parameters to increase accuracy or help convergence.

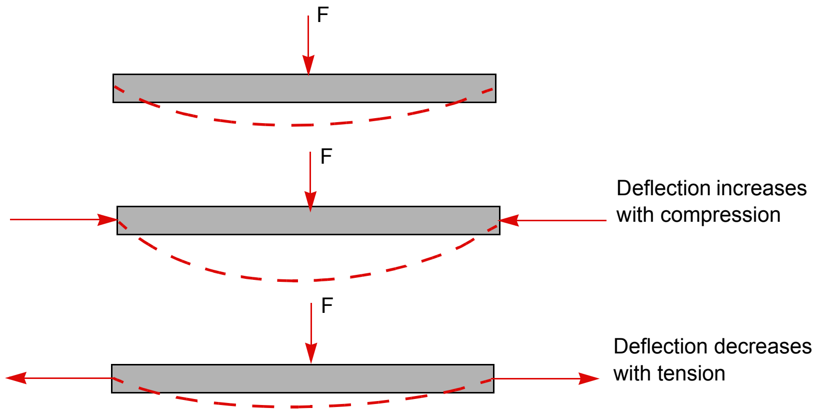
To control FFEPlus convergence parameters:

1. While the solver is running, click **PCG_Reset**. The solver stops and the *Reset PCG Parameters* window opens. In addition to the convergence parameters, the window lists the current iteration number and stopping threshold.
2. To modify convergence parameters, type in the desired values in the appropriate fields and click **OK**. The program continues using the parameters you specified. You can repeat the process as needed.

A large tolerance gives you a quick solution but less accurate results. We recommend that you do not trade off accuracy for speed unless convergence become extremely slow. In such cases, you can increase the tolerance slightly to help convergence.

Use Inplane Effect (Direct Sparse and FFEPlus solvers only)

Compressive and tensile loads change the capacity of a structure to resist bending. Compressive loads decrease resistance to bending. This phenomenon is called *stress softening*. Similarly tensile forces increase bending stiffness. This phenomenon is called *stress stiffening*.



Check this option to consider the effect of loads on the stiffness of the model. COSMOSWorks runs the static analysis twice to consider this effect.

An accurate solution for considering the effect of loads on the stiffness (capacity of resisting loads) requires nonlinear analysis.

Use Soft Spring to Stabilize Model (Direct Sparse and FFEPlus solvers only)

Check this option to instruct the program to add soft springs attached to the ground to prevent instability. If you apply loads to an unstable design, it will translate and/or rotate as a rigid body. You should apply adequate restraints to prevent rigid body motion.

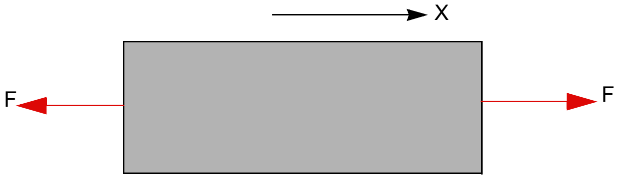
If you cannot stabilize the design by applying adequate restraints, turn on this flag and rerun the analysis. Animating the resulting deformed shape will show you excessive motion in one or more directions. You can then apply additional restraints to prevent rigid body motion. Repeat the ananalysis without activating this flag after fixing the problem.

In general, you should not activate this flag unless you ran the analysis and the program tells you that the constraints are not adequate, or you know that you have not specified adequate restraints.

This option is automatically applied for contact problems solved by the Direct Sparse solver.

Use Inertial Relief

If a model is not adequately restrained, a small force can cause rigid body motion. Even in cases when the applied forces are balanced, a small unbalanced force can result from numerical approximations.



Numerical approximations can lead to rigid body motion in the X direction even when the applied forces are balanced.

When this option is checked, the program automatically applies forces to counteract unbalanced external loading. This option is particularly useful when you import loads from COSMOSMotion where external loads can be slightly unbalanced. When you check this option, the program does not complain if the restraints are not adequate.

Do not check this flag if the external loads are not approximately balanced.

Adaptive Methods

Adaptive methods are based on error estimation. There are mainly two methods to improve the accuracy of the FEA results:

The H-Method

The concept of the h-method is to use smaller elements in regions with high errors. After running the study and estimating errors, you can use mesh control to specify smaller element sizes in regions with high errors. You can continue this process until you are satisfied with the level of accuracy. To estimate the errors, plot the Error stress component for elements (not nodes). This method is not automated in this release.

The P-Method

The concept of the p-method is to use more efficient elements in regions with high errors. After running analysis and estimating errors as outlined above, the program increases the order of elements in regions with errors higher than a user-specified level and reruns the study. The p-method does not change the mesh. It changes the order of the polynomials used to approximate the displacement field. Using a unified polynomial order for all elements is not efficient. COSMOSWorks increases the order of the polynomial only where it is needed. This approach is called the *selective* adaptive p-method

To improve the accuracy of the solution, check **Use p-Adaptive for solution** option to use the p-method. This option is supported for solid elements only, it is not supported for shell elements. When this option is checked, the program may run the problem several times. After each loop, the program assesses the global and local errors and decides whether to make another run.

The program stops the loops when one of the following conditions is met:

- the global criterion converges,
- all local errors converge (i.e. for each element), or
- the maximum number of loops is reached.

You can base the convergence check on total strain energy, Root Mean Square (RMS) of von Mises stresses, or RMS of resultant displacements.

Limitation: In this release, the p-method does not work when nonuniform pressure, nonuniform forces, or multiple pressures are defined on a face.

After running a static problem using the p-adaptive method, you can generate convergence plots. For more information, refer to the *Viewing Results* chapter. For an example, click the **COSMOSWorks Online Tutorial**.

Properties of Frequency Studies

The **Frequency** dialog box sets analysis properties for frequency studies. The dialog box has two tabs.

- The **Options** tab sets frequency analysis options.
- The **Remark** tab allows you to attach a remark to the study and include it in the study report.

Number of Frequencies

Lets you set the desired number of natural (resonant) *frequencies* to be calculated. The default is to calculate the lowest five frequencies. Rigid body modes are calculated by the FFE and FFEPlus solvers. A body without any restraints has six rigid body modes. Rigid body modes have zero frequencies (infinite period).

If the working scenarios of the model include dynamic loads, it is important to calculate at least one frequency that is higher than the frequency of the load. In most situations, resonance is not desirable because it causes failure. However, some devices exploit resonance to trigger an event while providing measures to control the excessive associated deformation.

Upper Bound Frequency

Lets you set the upper bound frequency of the desired frequency range. Use the default value of zero unless you are not interested in natural frequencies higher than a certain value. Inputting zero results in calculating the specified number of frequencies.

You must specify the **Number of Frequencies** or the **Upper Bound Frequency**.

Use Inplane Effect

Loads affect the modal characteristics of a body. For example, compressive loads decrease resonant frequencies and tensile loads increase them. This fact is easily demonstrated by changing the tension on a violin string. The higher the tension, the higher the frequency (tone). This option is not available for the FFE and FFEPlus solvers.

Option Unchecked

COSMOSWorks ignores the effect of loads on resonant frequencies and mode shapes.

Option Checked

COSMOSWorks considers the effect of loads on resonant frequencies and mode shapes. The calculations take more time because COSMOSWorks runs static analysis before running frequency analysis.

Use Soft Spring to Stabilize Model

Check this option to add soft springs to stabilize inadequately supported models. See **Properties of Static Studies** for more information.

The FFE and FFEPlus solvers do not require a stable model to calculate resonant frequencies. They include rigid body modes automatically. For example, if you ask for three modes and there are six possible rigid body modes, the solvers calculate nine modes. The first six modes are rigid body modes. The frequency of a rigid body mode is zero.

Properties of Buckling Studies

The **Buckling** dialog box sets analysis properties for buckling studies. The dialog box has two tabs.

- The **Options** tab sets buckling analysis options.
- The **Flow/Thermal Effects** tab sets parameters related to importing temperatures or pressure loads from COSMOSFloWorks or temperatures from a thermal study. Refer to *Properties of Static Studies* for details.

- The **Remark** tab allows you to attach a remark to the study and include it in the study report.

Like static analysis, you must prevent rigid body modes by applying adequate restraints or using the soft spring option. Buckling analysis is recommended for slender models with axial loads because failure due to buckling can occur at a load level that is smaller than that required to cause material failure due to high stresses.

Number of Buckling Modes

In most cases, only the lowest load factor is needed. However, the Direct Sparse solver allows you to request a number of buckling modes (buckling load factors and associated mode shapes).

> FFEPlus calculates the lowest buckling mode only. FFEPlus is only recommended for very large models. For most problems the Direct Sparse solver is more efficient.

The buckling load factor

The buckling load factor (BLF) is the factor of safety against buckling or the ratio of the buckling loads to the applied loads. The following table illustrates the interpretation of possible BLF values:

BLF Value	Buckling Status	Notes
1 < BLF	Buckling not predicted	The applied loads are less than the estimated critical loading.
0 < BLF < 1	Buckling predicted	The applied loads exceed the estimated critical loads.
BLF = 1	Buckling predicted	The applied loads are exactly equal to the estimated critical loading.
BLF = -1	Buckling not predicted	Buckling will be predicted if you *reverse* all loads. For example, if you apply a tensile force on a bar, BLF should be negative.
-1 < BLF < 0	Buckling not predicted	Buckling will be predicted if you *reverse* all loads.
BLF < -1	Buckling not predicted	Buckling will NOT be predicted even if you *reverse* all loads.

Calculating Buckling Loads

To calculate the buckling load(s) for a mode, multiply all applied loads by the BLF for that mode.

Use Soft Spring to Stabilize Model

Check this option to add soft springs to stabilize inadequately supported models. See **Properties of Static Studies** for more information.

Properties of Thermal Studies

The **Thermal** dialog box sets analysis properties for thermal studies. The dialog box has two tabs.

- The **Options** tab sets thermal analysis options.
- The **Remark** tab allows you to attach a remark to the study and include it in the study report.

Steady State and Transient Studies

A body is at a steady state if every point in the body is in thermal equilibrium. The temperature of each point remains unchanged. Use the steady state option to find the thermal equilibrium. Effects of initial conditions disappear at the steady state. This analysis does not tell you how long it takes to reach thermal equilibrium. You must define a heat dissipation mechanism like prescribed temperature or convection, otherwise thermal equilibrium may not be possible.

Transient studies calculate the thermal status of a body as a function of time. Based on a specified time increment (Δt), the program calculates the solution at Δt, $2(\Delta t)$, $3(\Delta t)$, $4(\Delta t)$,..., T, where T is the specified total time. You can graph the results at selected locations as a function of time. Initial temperatures are important for transient studies.

Properties of Optimization Studies

The **Optimization** dialog box sets properties for optimization studies. It has two tabs.

- The **Options** tab sets the maximum allowable number of optimization loops.
- The **Remark** tab allows you to attach a remark to the study and include it in the study report.

Maximum no. of design cycles

Optimization studies are based on associate studies of other types.

The optimization program stops when it reaches the maximum allowable number of design cycles, or when it finds the optimum design within the specified parameters.

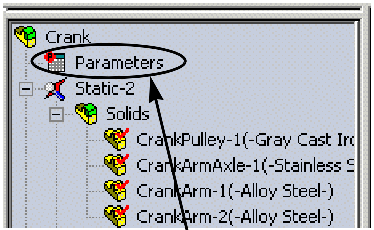
In each cycle, the program runs all studies used to specify the behavior constraints and the objective function. When running a study, the program uses the current properties associated with the study. For meshing, the program uses the active mesh properties for all studies.

Multiple Studies

You can create multiple studies as desired. Each study is presented in the COSMOSWorks Manager tree. When you run a study, the program uses the active mesh. You can use the same mesh for multiple studies with the same mesh type and default contact conditions (bonded). Contact options, other than **Bonded**, are used for static and thermal studies of solid assemblies only.

Parameters

You can link a numeric field to a parameter. A parameter is defined by name, type, unit, and a value or equation. A parameter icon appears in the COSMOSWorks Manager tree automatically. A parameter can be linked to define a model dimension, material property, element size, load, or restraint.



Parameters icon

Once you define a parameter of a certain type, you can use it to define input of associated fields. For example, if you define **Force1** as a force parameter, you can link it to a force input field instead of entering a numeric value. Changing the value of **Force1**, automatically changes the force linked to it.

To define parameters, double-click the **Parameters** icon in the COSMOSWorks Manager tree.

After you define your parameters, you can link one or more parameters to related input fields in your study.

To link a parameter to related input field, right-click inside the input field and select **Link Values** then select the desired parameter from the parameter list.

Design Scenarios

For static, frequency, buckling, and thermal studies, you can define up to 100 design scenarios based on parameter values.

Due to the size of the disk space required to save all results for all scenarios, the program saves detailed results for one design scenario and summary results for all other scenarios. Summary results include extreme values for selected quantities based on the study type as shown in the following table:

Associate study	**Results for listing and graphing**
Static	Maximum von Mises Stress Maximum equivalent strain Maximum principal stress Maximum resultant displacement von Mises stress, principal stresses, equivalent strain, and resultant displacement at up to 100 (vertices)
Frequency	Resonant frequencies
Buckling	Buckling load factors
Thermal	Maximum and minimum temperatures Maximum and minimum resultant heat gradients Maximum and minimum heat flux Temperature, resultant heat gradients, and heat flux at up to 25 locations (vertices)

When you run a study with defined design scenarios, you can run the design scenarios, or you can run the study as usual based on the current values of linked parameters.

If you run the design scenarios, COSMOSWorks evaluates the results for each active design scenario you defined.

When running the study for a certain design scenario, COSMOSWorks overwrites existing results based on the previous run. At the end of the analysis, the results and other folders of the associate study correspond to the last evaluated design scenario.

After completing the analysis, you can list and graph selected results.

Defining Design Scenarios

In order to define design scenarios for a study, you must first define at least one parameter and link it to the study.

When you create a static, frequency, buckling, or thermal study, COSMOSWorks creates a *Design Scenarios* icon in the COSMOSWorks Manager tree. To define analysis scenarios, right-click the **Design Scenarios** icon and select **Edit/Define**. You can define up to 100 design scenarios.

Limitation

If flipping is required to align shell faces for models created with the *Shell using midsurfaces* option, then nodal stress results may not be accurate since flipping is not performed when solving design scenarios. Other results like element stresses, displacements, element strains, and temperatures (for thermal studies) are not affected. Frequency and buckling studies do not require shell alignment. This problem does not appear in shell models created with the *Shell using surfaces* option as flipping is performed in each design scenario if it is performed in the initial study.

Running Studies

Verifying the Input

It is important to verify your input before running a study:

- Verify that you have assigned the proper material for each component/shell.
- Verify that you have specified the proper study properties.
- Verify that you have specified the proper loads and restraints.
- If you used parameters to define design scenarios, then verify the values of the parameters and the linked fields.
- Verify the mesh and make sure it corresponds to the desired mesh options.

Running the Study

When you run a study, COSMOSWorks calculates the results based on the geometry, material, loads and boundary conditions, and mesh. For studies with design scenarios, you can run the study as usual based on the current values of the linked parameters, or you can run the design scenarios.

You can choose to run a study automatically after meshing it by checking the **Run analysis after meshing** option in the **Mesh** PropertyManager. For studies with defined design scenarios, this option runs the study based on the current linked parameter values only. It does not run the design scenarios.

To obtain summary results for all active design scenarios and detailed results for the last active design scenario, right-click the study icon in the COSMOSWorks Manager tree and click **Run Design Scenarios**.

To ignore the design scenarios and calculate detailed results based on the current values of linked parameters, right-click the study icon in the COSMOSWorks Manager tree and click **Run**.

Exporting Studies

You can export a study to other FEA programs. In addition to COSMOS GEOSTAR, you can export your study to ANSYS, NASTRAN, PATRAN, I-DEAS, and Exodus. To set export preferences, click **COSMOSWorks, Preferences** and click the **Export** tab.

The GEOSTAR translator is provided with the package. For other programs, you need to acquire the corresponding translator as an add-on for this function to work.

To export a study, right-click the study icon in the COSMOSWorks Manager and click **Export**. The following table lists the files that you can generate:

FEA Program	**File Extension**
GEOSTAR	GEO
ANSYS	ANS (Prep 7 file)
MSC NASTRAN	DAT
PATRAN	NEU (Neutral file)
I-DEAS	UNV (Universal file)
Exodus	TXT

4

Material Properties

This chapter discusses topics related to assigning material properties to the model. The following topics are discussed:

- **Ways of Defining Material Properties**
- **Material Models**
- **Material Properties Used in COSMOSWorks**
- **COSMOS Material Browser**

Before running a study, you must define all the necessary material properties required by the corresponding analysis type. For example, the modulus of elasticity is required for static, frequency, and buckling studies, while thermal conductivity is needed for thermal studies. You can define material properties at any time before running the analysis. All material properties are defined through the **Material** dialog box.

When you create a study, the program creates a **Solids** or **Shells** folder (depending on the selected mesh type option), a **Load/Restraint** folder, and a **Design Scenario** icon in the COSMOSWorks Manager tree under the study. A **Contact/Gaps** icon is also created for assembly documents. If you assigned a material to a part in SolidWorks, COSMOSWorks uses the same material automatically when you create a study. The material name assigned to the part appears in the COSMOSWorks Manager tree.

If you define a study using solid mesh for a part, the **Solids** folder will contain one icon in it. If you define a study using solid mesh for an assembly, the **Solids** folder will contain an icon for each component. For studies created with the **Shells using surfaces** option, an icon is created for each shell you define.

Ways of Defining Material Properties

You can define material properties in the following ways:

- ❑ Use materials assigned to parts in SolidWorks (default)
- ❑ Pick a material from the COSMOS Material Library
- ❑ Define material properties manually
- ❑ Pick a material from the Centor Material Library (an add-on option)

You can use the *COSMOS Material Browser* to add more materials or create new material libraries. You can also add materials to the SolidWorks material library.

The Material Dialog Box

Use the **Material** dialog box to define material properties for solid components and shells.

To access the Material dialog box:

- Right-click the **Solids** or **Shells** folder in the COSMOSWorks Manager tree and select **Apply Material to All**.
- Right-click one or more components in the **Solids** folder or a shell icon in the **Shells** folder in the COSMOSWorks Manager tree and select **Apply/Edit Material**.
- Click **COSMOSWorks, Apply/Edit Material** to assign a material to all components or shells.
- Drag and drop or cut and paste from one study to another.

Material Models

A material model describes the stress-strain relation for a material. A linear elastic material model describes the elastic behavior of a material in the linear range. The Material dialog box offers two types of material models:

- Linear Elastic Isotropic
- Linear Elastic Orthotropic

Assumptions of Linear Elastic Material Models

Linear elastic material models make the following assumptions:

Linearity Assumption

The induced response is directly proportional to the applied loads. For example, if you double the magnitude of loads, the model's response (displacements, strains, and stresses) will double. You can make the linearity assumption if the following conditions are satisfied:

- The highest stress is in the linear range of the stress-strain curve characterized by a straight line starting from the origin. As the stress increases, materials demonstrate nonlinear behavior above a certain stress level. This assumption asserts that the stress should be below this level. Some materials, like rubber, demonstrate a nonlinear stress-strain relationship even for low stresses.
- The maximum displacement is considerably smaller than the characteristic dimension of the model. For example, the maximum displacement of a plate must be considerably smaller than its thickness and the maximum displacement of a beam must be considerably smaller than the smallest dimension of its cross-section.

Elasticity Assumption

The loads do not cause any permanent deformation. In other words, the model is assumed to be perfectly elastic. A perfectly elastic model returns to its original shape when the loads are removed.

Isotropic and Orthotropic Materials

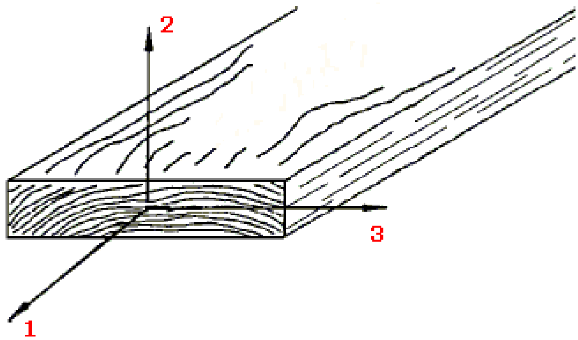
Isotropic Materials

A material is isotropic if its mechanical and thermal properties are the same in all directions. Isotropic materials can have a homogeneous or non-homogeneous microscopic structures. For example, steel demonstrates isotropic behavior, although its microscopic structure is non-homogeneous.

Orthotropic Materials

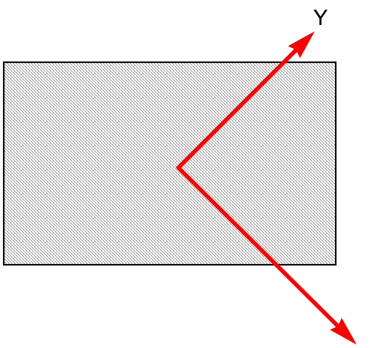
A material is orthotropic if its mechanical or thermal properties are unique and independent in three mutually perpendicular directions. Examples of orthotropic materials are wood, many crystals, and rolled metals.

For example, the mechanical properties of wood at a point are described in the longitudinal, radial, and tangential directions. The longitudinal axis (1) is parallel to the grain (fiber) direction; the radial axis (2) is normal to the growth rings; and the tangential axis (3) is tangent to the growth rings.

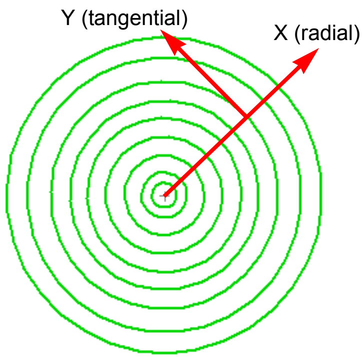


Orthotropic Directions

Orthotropic directions are defined with respect to a reference plane, axis, or coordinate system. For example, if you select an axis as a reference, X defines the radial direction, Y defines the tangential direction, and Z defines the axial direction.



Material properties defined with respect to a plane or coordinate system



Material properties defined with respect to an axis

Material Properties Used in COSMOSWorks

Elastic Modulus

Elastic Modulus in the global X, Y, and Z directions. For a linear elastic material, the elastic modulus in a certain direction is defined as the stress value in that direction that causes a unit strain in the same direction. Also, it is equal to the ratio between the stress and the associated strain in that direction. The modulus of elasticity was first introduced by *Young* and is often called **Young's Modulus**.

Elastic Moduli are used in static, frequency, and buckling analyses.

Shear Modulus

The *shear modulus*, also called modulus of rigidity and modulus of elasticity in shear, is the ratio between the shearing stress in a plane divided by the associated shearing strain.

Shear Moduli are used in static, frequency, and buckling analyses.

Poisson's Ratio

Extension of the material in the longitudinal direction is accompanied by contractions in the lateral directions. If a body is subjected to a tensile stress in the X-direction, then *Poisson's Ratio* **NUXY** is defined as the *ratio of lateral contraction in the Y-direction divided by the longitudinal strain in the X-direction*. Poisson's ratios are dimensionless quantities. For isotropic materials, the Poisson's ratios in all planes are equal (NUXY= NUXZ = NUYZ).

Poisson ratios are used in static, frequency, and buckling analyses.

Coefficient of Thermal Expansion

The *Coefficient of Thermal Expansion* is defined as *the change in length per unit length per one degree change in temperature* (*change in normal strain per unit temperature*).

Coefficients of thermal expansion are used in static, frequency, and buckling analyses if thermal loading is used. Frequency analysis uses this property only if you consider the effect of loads on the frequencies (in-plane loading).

Thermal Conductivity

The *Thermal Conductivity* indicates the effectiveness of a material in transferring heat energy by conduction. It is defined as the rate of heat transfer through a unit thickness of the material per unit temperature difference.The units of thermal conductivity are Btu/in sec ^{o}F in the English system and W/mK in the SI system.

Thermal conductivity is used in transient thermal analysis.

Density

The *Density* is mass per unit volume. Density units are lb/in^3 in the English system and kg/m^3 in the SI system.

Density is used in static, frequency, buckling, and thermal analyses. Static and buckling analyses use this property only if you define body forces (gravity and/or centrifugal).

Specific Heat

The *Specific Heat* of a material is the quantity of heat needed to raise the temperature of a unit mass of the material by one degree of temperature. The units of specific heat are Btu in/lbf^{o}F in English system and J/kg K in the SI system. This property is used in thermal analysis only.

COSMOS Material Browser

The **COSMOS Material Browser** provides a functionality to edit the COSMOS Material Library and create your own material libraries.

There are several classes of materials in the COSMOS Material Library: Steel, Iron, Aluminum, Copper etc. Each class has a number of materials. You can edit the COSMOS Material library by adding new materials in an existing class, creating new classes of materials, or changing the given values of material properties.

To start the browser, click **Start, Programs, COSMOS Applications, Material Browser**. Use the online help of the browser to learn its capabilities.

SolidWorks 2004 allows you to assign a material from SolidWorks material library to a part. You can add your own materials to SolidWorks material library. Refer to the *SolidWorks Online Help* for details.

5

Loads and Restraints

This chapter discusses the application of loads and restraints in COSMOSWorks. You learn about the following topics:

- **Using Reference Geometry**. Learn about using reference planes and axes to define directional loads and restraints.
- **Displacement Restraints**. Learn about applying displacement restraints for structural studies.
- **Structural Loads**. Learn about applying loads for structural studies.
- **Connectors**. Learn about defining connectors for structural studies.
- **Thermal Loads and Restraints**. Learn about applying loads and restraints for thermal studies.
- **Thermal Loads and Restraints.** Learn about applying loads and restraints for studies meshed by shell elements.
- **Miscellaneous Examples**. See some examples of applying loads and restraints.

Loads and restraints are necessary to define the service environment of the model. The results of analysis directly depend on the specified loads and restraints. Loads and restraints are applied to geometric entities as features that are fully associative to geometry and automatically adjust to geometric changes.

For example, if you apply a pressure **P** to a face of area **A_1**, the equivalent force applied to the face is **PA_1**. If you modify the geometry such that the area of the face changes to **A_2**, then the equivalent force automatically changes to **PA_2**. Remeshing the model is required after any change in geometry to update loads and restraints.

When you create a study, the program creates a **Load/Restraint** folder in the COSMOSWorks Manager tree. COSMOSWorks adds an icon in the **Load/Restraint** folder each time you define a load or restraint on one or more entities.

The types of loads and restraints available depend on the type of the study. A load or restraint is applied by the corresponding PropertyManager accessible by right-clicking the **Load/Restraint** folder of a study in the COSMOSWorks Manager tree, or by clicking **COSMOSWorks**, **Insert**. Other types of structural loads are available via special functions like importing loads from COSMOSFloWorks or COSMOSMotion.

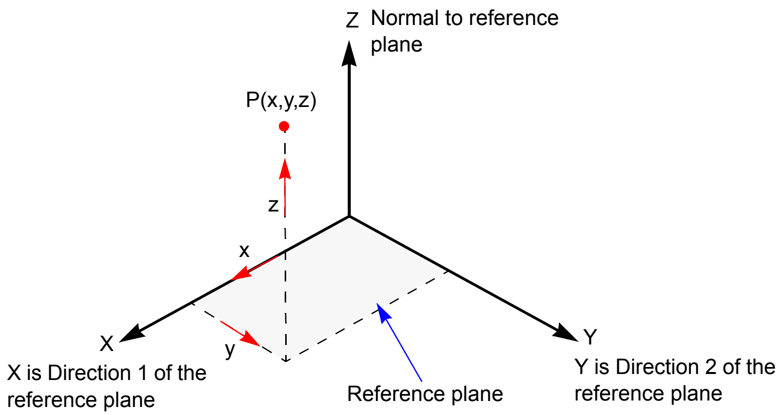
To help you define studies faster, you can drag and drop **Load/ Restraint** folders and items from one study to another compatible study in the COSMOSWorks Manager tree. You can also copy a study and other compatible folders and items.

Using Reference Geometry

When applying a load or restraint condition, you need to specify a direction. COSMOSWorks provides options that are sensitive to the selected entities to which you want to apply the condition. In some cases, you need to use a reference plane or axis to specify the direction of the load or restraints condition. COSMOSWorks refers, by default, to the global coordinate system. The global coordinate system is based on Plane1 with its origin located at the Origin Origin of the part or assembly. Plane1 is the first plane that appears in the tree and can have a different name. You can use other reference planes or axes to specify directions.

Using Reference Planes

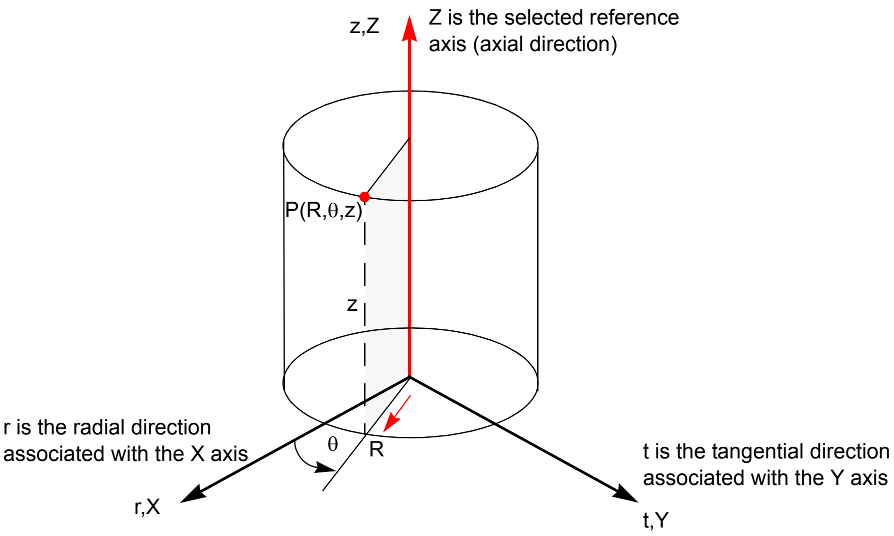
A reference plane defines a Cartesian coordinate system as illustrated in the figure.



Cartesian Coordinate System

Using Reference Axes

A reference axis defines a cylindrical coordinate system as illustrated in the figure. Reference axes are useful and necessary for specifying radial and tangential restraints.



Cylindrical Coordinate System

Displacement Restraints

In many cases, the displacements at certain locations are known. Displacement restraints are required to prevent rigid body motion. You can apply zero or nonzero displacement constraints to vertices, edges, and faces. When fixing an entity in all directions, there is no need to refer to a reference system. However, when the motion of a vertex, edge, or face is only known in a specific direction, you can select a plane or an axis to specify the direction.

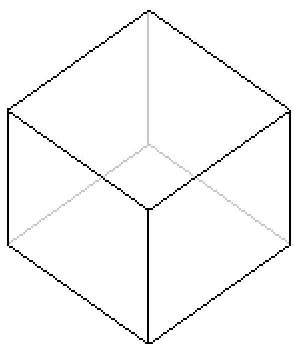
Displacement restraints are used in static, buckling, and frequency studies. The following restraints are available for structural studies:

- Prescribed displacements on vertices, edges, and faces
- Remote restraints
- Restraints based on defined connectors (rigid, spring, pin, and elastic support)

To prevent rigid body motion in static and buckling studies, you must apply adequate restraints, or use soft springs to stabilize the model. For frequency studies, displacement restraints are NOT required if you use the FFE or FFEPlus solvers.

Adequate Restraints for Solid Models

Displacement restraints, connectors, contact conditions, and study properties define the stability of the model. Restraints are adequate, if they do not allow any rigid body motion of the whole model or any of its components. For example, consider a cube meshed as a solid:



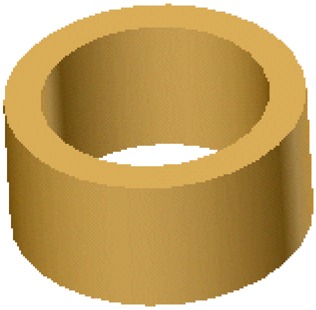
- If you fix one vertex, the model is not stable because it can rotate around the fixed vertex.
- If you fix two vertices, the model is not stable because it can rotate about the line connecting the two vertices.
- If you fix an edge, in general a *straight* edge, the model is not stable because it can rotate about the fixed edge.
- If you fix an edge and restrain a face that is normal to the edge from motion in the normal direction, the model is not stable because it can rotate about the edge.
- If you fix three vertices, the model is stable.

In general, fixing any number of vertices along a straight line is not adequate to stabilize a model.

- If you fix a face, the model is stable.

- If you fix an edge and a vertex that is not part of the edge, the model is stable.
- If you restrain three faces that are normal to each other from motion in the normal direction, the model is stable.

For another example, consider a hollow cylinder:



- If you restrain a cylindrical face in the radial direction, the model is not stable because it can rotate and slide.
- If you restrain a cylindrical face in the tangential directions, the model is not stable because it can slide along its axis.
- If you restrain a cylindrical face in the radial and tangential directions and a flat face in the axial direction, the model is stable.
- If you restrain a cylindrical face in the tangential direction and a flat face in the axial direction, the model is stable.
- If you fix any face, the model is stable.

Adequate Restraints for Shells

Shell elements consider translations as well as rotations at their nodes. The **Immovable** and **Fixed** restraint conditions are similar for solid elements but are different for shell elements. **Immovable** sets translations to zero but does not restrain rotations. **Fixed** sets all translations and rotations to zero.

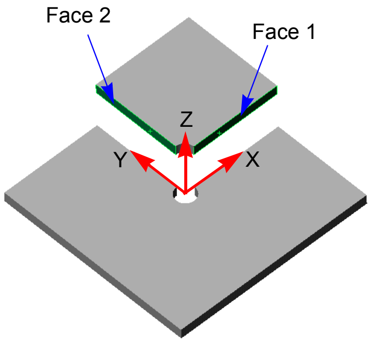
- Consider a plate meshed with shell elements.
- If you make a vertex or edge immovable, the model is not stable because it can rotate about this vertex.
- If you make an edge immovable, the model is not stable because it can rotate about this edge.
- If you fix an edge or more than one vertex, the model is stable.
- If you fix one vertex, the model may or may not be stable depending on the applied loads.

Symmetrical Restraints

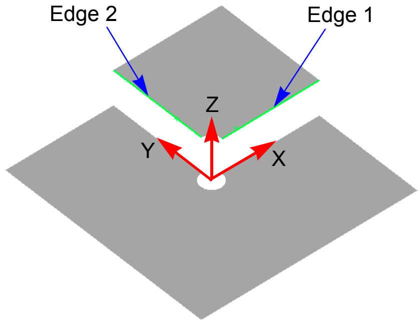
For solid elements, a face of symmetry is specified by setting normal translation to zero. For shell models, symmetry requires setting a translation and two rotations to zero.

For example, consider the plate shown in the figure. Due to double symmetry, you can model one quarter of the plate. If you use a solid mesh, you must set the translations normal to faces 1 and 2 to zero. If you use a shell mesh, you need to apply the following boundary restraints:

- On *Edge 1*: Set the y-translation and the x- and z-rotations to zero.
- On *Edge 2*: Set the x-translation and the y- and z-rotations to zero.



Symmetry restraints for the solid model of a plate

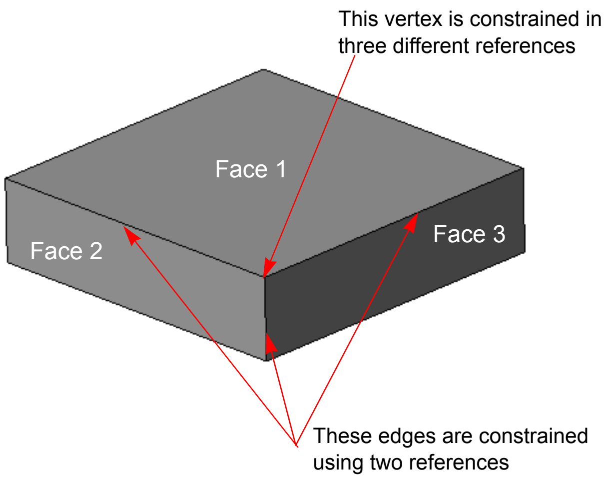


Symmetry restraints for the shell model of a plate

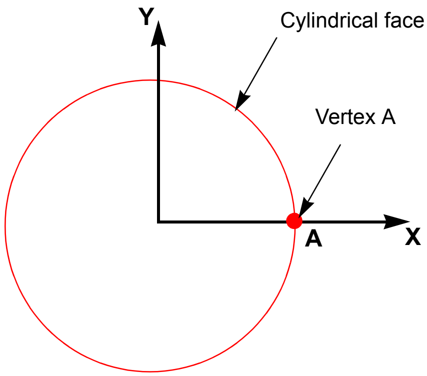
- For thermal studies, symmetry indicates insulation. Since a face or an edge without any applied boundary conditions is considered insulated, symmetry is indicated if no boundary conditions are applied.

Multiple Application of Displacement Restraints

Usually one reference is needed to specify the motion at a location. However, a location can receive multiple assignments when it is common to higher entities with prescribed restraints. For example, suppose that you applied symmetry condition to faces 1, 2, and 3 shown in the figure below. The three edges that are common to faces will receive restraints in two references. The common vertex will receive restraints in three different references. COSMOSWorks enforces all specified restraints. If it finds a contradiction, it will give a message before stopping the analysis.



As an example, if you fix Face 1 completely and prescribe a normal motion of 0.01” at Face 2, a contradiction arises at the common edge. Another example is illustrated in the following figure.



Suppose that vertex A is constrained (no motion) in direction 1 of Plane1 (global X), and the cylindrical face is given a radial displacement of 0.1”. Obviously there is a contradiction at vertex A. The analysis program will give a message and stop. However, if the tangential rotation of the face is set to a value while the radial motion is prevented, there is no contradiction and the analysis continues.

Summary of Displacement Restraint Options

Restraints options are sensitive to the selected geometric entities, planes, and axes. The table below lists possible restraints and associated entity types:

Restraint Type	Geometric Entities	Reference Geometry Type	Required Input
Fixed (fixes translations and rotations)	Vertices, edges, and faces	N/A	N/A
Immovable (fixes translations only)	Vertices, edges, and faces	N/A	N/A
Reference plane or axis	Vertices, edges, and faces	Axis or Plane	Prescribed translations and rotations in the desired directions
On flat face	Planar faces	N/A	Prescribed translations and rotations in the desired directions associated with the selected planar face
On cylindrical face	Cylindrical faces	N/A	Prescribed translations and rotations in the desired directions associated with the selected cylindrical face
On spherical face	Spherical faces	N/A	Prescribed translations and rotations in the desired directions associated with the selected spherical face
Remote restraint for static studies only	Faces	Coordinate System or default Plane1	Prescribed translations and rotations in the desired directions
Rigid connector	Faces	N/A	Two sets of faces belonging to Two different components

Structural Loads

Some type of loading, or a prescribed displacement, is required for static and buckling studies. Loads are optional for frequency studies and are used only if inplane loading flag is checked in the properties of the frequency study.

The following types of loads are available for structural studies:

- **Pressure** (uniform or nonuniform distribution)
- **Force** (uniform or nonuniform intensity)
- **Gravity**
- **Centrifugal Loads**
- **Remote Loads** (direct load transfer or rigid connection)
- **Bearing Loads**
- **Temperature** (prescribed temperatures, uniform temperature change, or a temperature profile from a thermal study)
- **Shrink Fitting** (applied as a contact condition)

COSMOSWorks can import loads from COSMOSMotion as well as pressure and temperatures from COSMOSFloWorks.

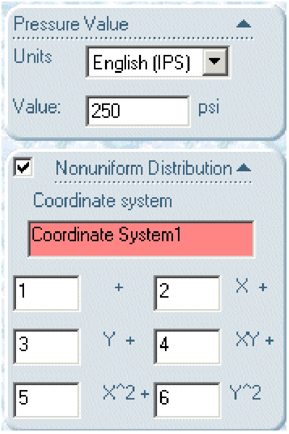
Pressure

The **Pressure** PropertyManager applies uniform or nonuniform (variable) pressure to faces for use in structural studies. Uniform pressure is applied in the specified direction with uniform distribution to all selected faces. Pressure can be applied normal to the selected faces or it can be applied in some other direction. For example, hydrostatic pressure is normal to faces while the snow on a sloped roof applies a vertical pressure and a wind blowing horizontally applies a horizontal pressure.

The equivalent force magnitude generated by pressure is equal to the pressure value times the area of the face. However, the net equivalent force depends on the geometry of the face and the direction of the pressure. For example, the reaction force resulting from applying a hydrostatic pressure on a full cylindrical face is zero due to symmetry.

Nonuniform pressure is described by a multiplier and a pressure distribution. The pressure distribution is described by the coefficients of a second-order polynomial in terms of a reference coordinate system.

The coordinate system should be oriented such that the distribution on the target face varies with respect to the x and y coordinates only.



This setting applies the following pressure:

$$P(x, y) = 250(1 + 2x + 3y + 4xy + 5x^2 + 6y^2)$$

The coefficients should be specified based on the unit of length (for x and y) as shown in the following table:

Selected Unit System	**Units of x and y**	**Units of "Value"**
SI	m (meters)	N/m^2
English (IPS)	in (inch)	psi
Metric (G)	cm (centimeters)	kilogram force/cm^2

Force

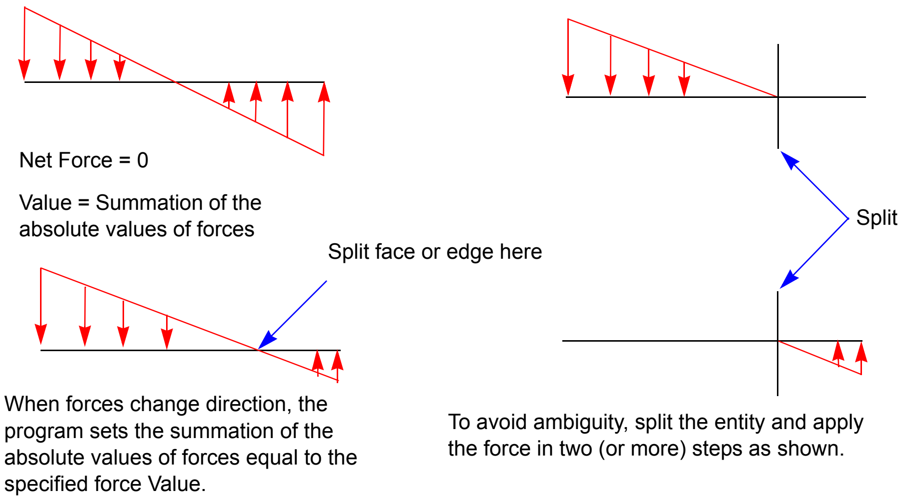
The **Force** PropertyManager applies forces, moments, or torques with uniform distribution to faces, edges, and vertices in any direction for use in structural studies. The specified force value is applied to EACH selected vertex, edge, and face.

You can apply nonuniform forces to faces and edges only. A nonuniform force is specified by a force value and force intensity. The force value is set equal to the summation of the absolute values of the forces applied to EACH face and edge. The force intensity is described by the coefficients of a second-order polynomial in terms of x and y axes of the reference coordinate system as described for variable pressure.

Variable torque is not supported in this release.

Distribution with Forces Changing Directions

If a distribution is specified such that forces change direction on part of the edge or a face as illustrated in the figure, the program sets the summation of the *absolute* values of the forces equal to the value specified in the PropertyManager.



Gravity

The **Gravity** PropertyManager applies linear accelerations to a part or assembly document for use in structural analyses. You specify accelerations in the x, y, and z axes of a reference plane. Gravity loading in each direction is calculated by multiplying the specified acceleration of gravity by the mass. The mass is calculated from the density value of the material. If you select materials from the COSMOSM or library, then the density is already defined. If you choose to input material properties manually, make sure to specify the density.

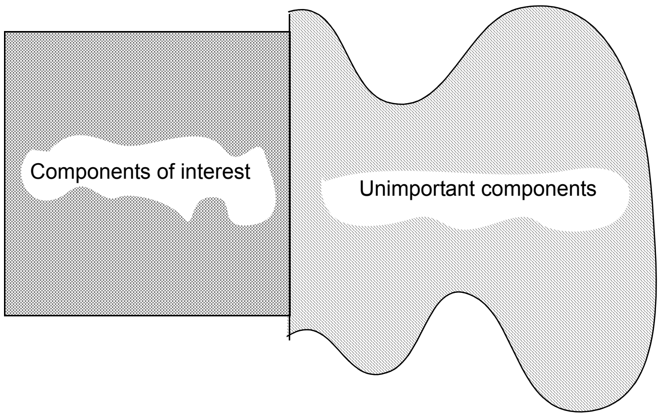
Centrifugal Loads

The **Centrifugal** PropertyManager applies angular velocity and acceleration about an axis to the whole part or assembly for use in structural analyses. The program uses the specified values and the mass density to calculate the centrifugal loads.

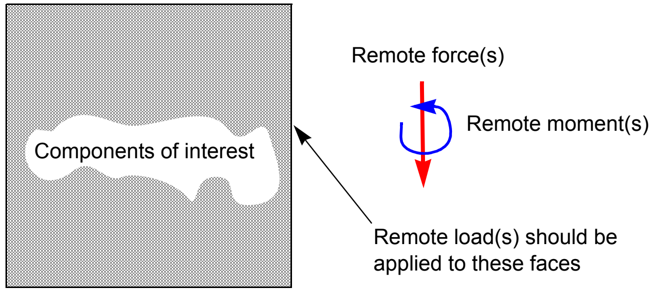
Remote Loads

In many situations, the local results of a component are not of interest. However, the effect of the component on the rest of the model cannot be neglected. The function of the component is to transfer the loads. To simulate the effect of such components on the rest of the model, you suppress the component and apply remote loads and/or restraints outside the model. This can lead to simplifying the model without much effect on the accuracy away from the interface of the neglected component with other components.

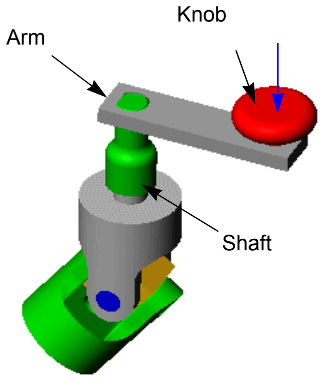
As the name suggests, remote loads and restraints are generally applied at remote locations (outside the model) and transferred to selected faces. This concept is illustrated in the following schematic figure.



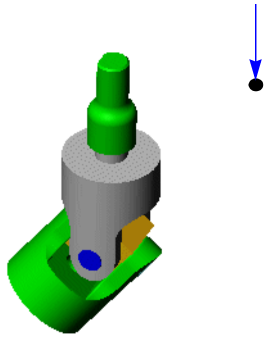
The above situation can be modeled using remote loads as follows.



The next figure illustrates how remote loading is used to replace a crank knob and arm in a joint assembly.



Force or pressure applied
to the crank knob



Remote force applied at the location of the Knob center. The force is applied to faces of the Shaft that interface with the Arm.

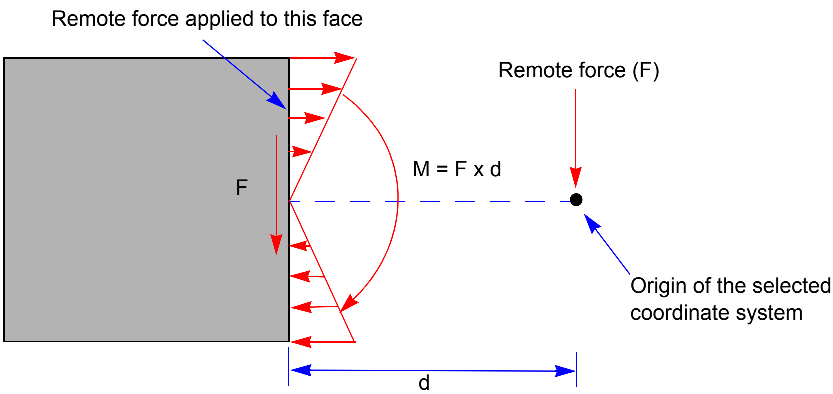
Types of Remote Loads/Restraints

There are three options for applying remote loads/restraints.

Load (Direct transfer)

You can use this option when the interface with the omitted component does not prevent the specified face(s) from deforming as flexible bodies. Remote loads (forces and/or moments) are applied at a point defined by the X-, Y-, and Z-locations in reference to a selected coordinate system. The global coordinate system is used by default if no coordinate system is selected. Remote moments are applied about the specified axes of the coordinate system. The program automatically calculates and applies equivalent forces to the selected faces. A force applied at the remote location transfers as a force, 2 equivalent moments, and a torque on the selected face(s).

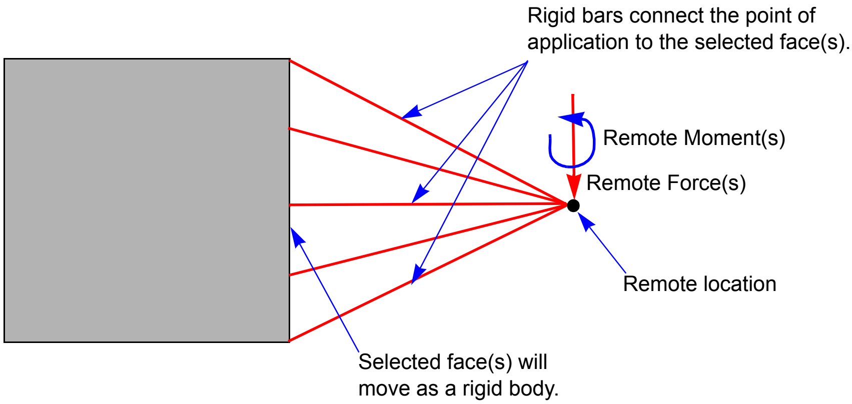
For example, if you apply a remote force F to a face as shown in the figure, the program applies forces that are equivalent to a moment M = F x d in addition to the force F.



Load (Rigid connection)

You can use this option when the replaced components are adequately rigid with respect to the modeled components. The selected faces can only deform as a rigid body. The area and shape of each face remain unchanged. High stresses can develop near faces with rigid connections.

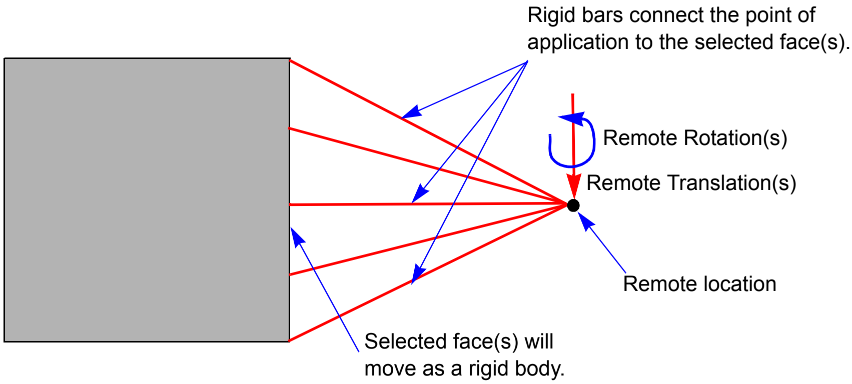
The load is transferred to the selected face(s) as if is the point of application of the remote load is connected to the selected faces by rigid bars.



Displacement (Rigid connection)

You can use this option when the replaced components are adequately rigid with respect to the modeled components and you know the remote translations and/or rotations that can replace its effect on the rest of the model. The point of application of the constraint is effectively connected to the selected faces by rigid bars.

The selected face(s), being rigidly connected to a common point, can only deform as a rigid body. The area and shape of each face remain unchanged. High stresses can develop near faces with rigid connections.

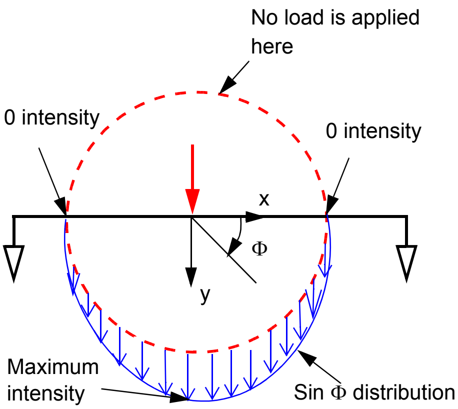


Bearing Loads

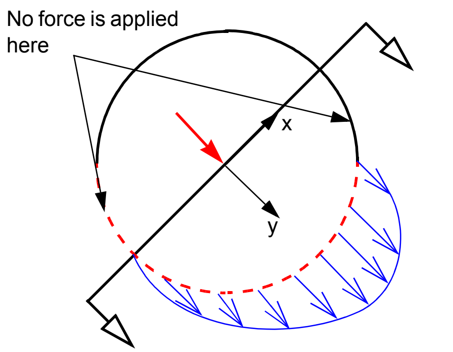
Bearing loads develop between contacting cylindrical faces. In most cases, the contacting faces have the same radius. The bearing forces generate a nonuniform pressure at the interface of contact. The program assumes a sine variation in the appropriate half-space as shown in the figure.

Illustration

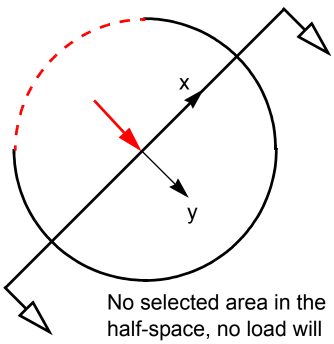
The half-space is defined by the space in which the angle Φ is less than or equal to 180^0.



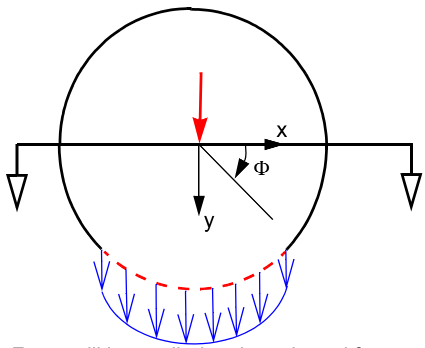
Force will be applied to the lower half of the selected face only



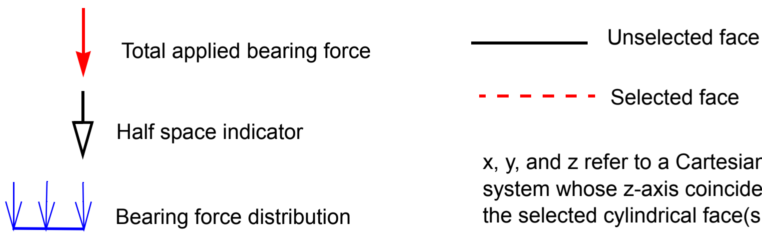
Force will be applied only to the portion of the selected face located in the half space



No selected area in the half-space, no load will be applied.



Force will be applied to the selected face

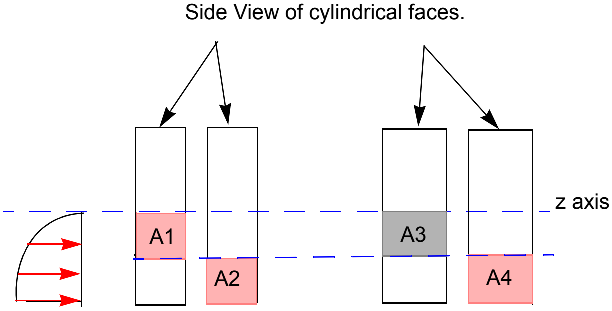


x, y, and z refer to a Cartesian coordinate system whose z-axis coincides with the axis of the selected cylindrical face(s)

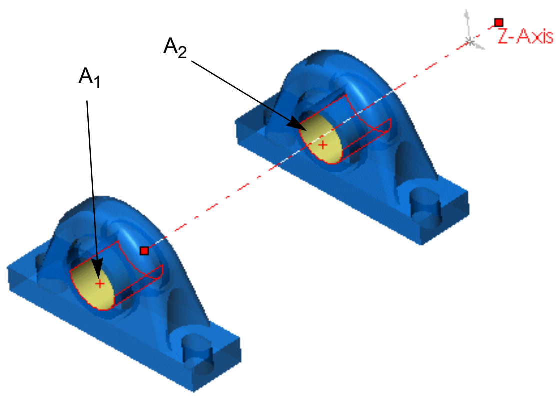
Applying Bearing Forces to Multiple Faces

When you apply a bearing force to multiple faces, COSMOSWorks divides the force among the selected faces based on their areas and locations in the half-space (where Φ is less than or equal to 180^0).

A bearing force is applied to selected cylindrical faces as schematically shown in the figure. Suppose that selected area A1 = A2 and A3 = A4. A2 will receive more force because of its maximum intensity location in the half-space. Similarly, A4 will get more force than A3. The ratio of the forces received by A1 and A3 is directly proportional to areas since they occupy the same relative positions in the half-space. The same thing can be said for A2 and A4.



As illustrated in the following figure, if the selected areas A1 and A2 in the half-space and their relative positions with respect to the half-space are identical, the force will be divided equally.



Importing Loads

COSMOSWorks imports loads from COSMOSMotion and COSMOSFloWorks.

Importing Loads from COSMOSMotion

Many products contain moving assemblies of components (mechanisms). Mechanisms play a crucial role in the performance of such products. COSMOSMotion is a software that simulates mechanical systems in SolidWorks.

After studying the mechanism in COSMOSMotion, you can export the loads generated by the specified motion to COSMOSWorks.

Refer to the *Online Help* for details. For an example, refer to the *COSMOSWorks Online Tutorial*.

Importing Pressure and Temperatures from COSMOSFloWorks

COSMOSFloWorks is a Computational Fluid Dynamics (CFD) software that allows you to analyze a wide range of fluid flow problems. Among the quantities it calculates are the pressure distribution and temperature profile induced by fluid flows. You can now import pressure and temperature results from COSMOSFloWorks to COSMOSWorks static and buckling studies.

Refer to the *Online Help* for details. For an example, refer to the *COSMOSWorks Online Tutorial*.

Shrink Fitting

Shrink fitting is encountered in many engineering designs. It refers to fitting a component into a slightly smaller cavity. Due to normal forces that develop at the interface, the inner component shrinks while the outer component expands. The amount of shrinkage/ expansion is determined by the material properties as well as the geometry of the components.

The contacting faces need not be cylindrical.

Shrink fitting is implemented as a contact condition. You can define a shrink fit contact condition using the **Shrink Fit** option in the **Contact Pair** PropertyManager. It should be noted that you need to remesh the model whenever you make a change in the contact conditions.

Illustration

The following figure shows an example of a situation in which the **Shrink Fit** option is used. In this example, a cylinder is compressed and then placed inside another cylinder. The **Shrink Fit** contact condition is applied to the contacting faces of the two cylinders (the outer face of the inner cylinder and the inner face of the outer cylinder). For more information on the **Shrink Fit** contact, refer to the *Meshing* chapter.

- For accurate results, the overlap should be large enough to overcome approximations introduced by meshing. For example, the overlap of cylindrical or spherical faces should be larger than 0.1% of the larger diameter at the interface for accurate results.
- This example is provided as step-by-step lesson in the COSMOSWorks Online Tutorial.

The geometry should be modeled as interfering. Local contact as shrink fit should be specified for the desired face pairs before meshing.

Summary of Structural Loads

Applying loads for structural studies is sensitive to the selected reference entity. The following table summarizes options available for applying structural loads.

Load Type	Geometric Entities	Reference Geometry Type	Required Input
Normal Force with *uniform* intensity	Faces of solids and shells, and edges of shells.	N/A	Unit and value of the force. The program applies the specified force to each face.

Load Type	**Geometric Entities**	**Reference Geometry Type**	**Required Input**
Normal Force with *nonuniform (variable)* intensity	Faces	Coordinate system	Unit and value of the force and polynomial coefficients to define the intensity. The program applies the specified force to each face.
Directional Force with *uniform* intensity	Vertices, Edges, and Faces	Axis, or Plane	Unit and value of the force. The program applies the specified force to each entity.
Directional Force with *nonuniform* intensity	Faces of solids and shells, and edges of shells.	Axis or plane for direction, and Coordinate system for intensity	Unit and value of the force and polynomial coefficients to define the intensity. The program applies the specified force to each entity.
Torque	Faces. Usually circular or cylindrical faces	Axis	Unit and value of the torque. The program applies the specified torque to each face.
Normal Pressure with *uniform* distribution	Faces of solids and shells, and edges of shells	N/A	Unit and value of the pressure.
Normal Pressure with *nonuniform* distribution	Faces of solids and shells, and edges of shells	Coordinate System to define the distribution	Unit and value of the pressure.and polynomial coefficients to define the distribution
Directional Pressure with *uniform* distribution	Faces of solids and shells, and edges of shells	Reference plane	Unit and pressure values in the desired directions

Load Type	Geometric Entities	Reference Geometry Type	Required Input
Directional Pressure with *nonuniform* distribution	Faces of solids and shells, and edges of shells	Plane to define direction, and coordinate system to define distribution	Unit and value(s) of the pressure in the desired direction(s) and polynomial coefficients to define the intensity
Gravity	Whole model	Plane	Unit and acceleration value(s). Material density must be defined
Centrifugal Forces	Whole model	Plane	Unit and angular velocity and/or acceleration value(s). Material density must be defined
Remote Load/ Restraint (Direct transfer or Rigid connection)	Faces	Coordinate system (default is Plane1)	Prescribed forces/ translations and moments/ rotations in the desired directions
Direct Bearing Loads	Cylindrical faces	Coordinate system	A force in the X or Y direction of the selected coordinate system Z axis of the coordinate system must coincide with the axis of the cylindrical face(s)
Shrink Fit	Faces	N/A	A pair of faces. Each face on a different component.
Temperature	Vertices, Edges, and Faces	N/A	Unit and temperature value.

Multiple Application of Structural Loads

You can apply forces and pressure as many times as desired. COSMOSWorks *superimposes* (accumulates) all pressures, forces, and remote loads. However, COSMOSWorks allows only one assignment for gravity loading and one assignment for centrifugal loading for a study.

Connectors

A connector is a mechanism that defines how a face is connected to another face or to the ground. Connectors are encountered in many real life designs. Using connectors simplifies modeling. In many cases, you can simulate the desired behavior without having to create the detailed geometry or define contact conditions.

The **Connectors** PropertyManager enables you to define the following types of connectors:

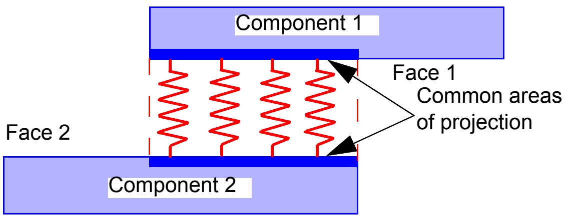
- **Rigid** (Rigid Connection in earlier releases)
- **Spring**
- **Pin**
- **Elastic support**

Rigid Connector

A rigid connector connects faces of distinct components to each other rigidly. The selected faces can only move as one rigid body. This functionality is not new. The **Rigid Connection** PropertyManager available in earlier releases of COSMOSWorks is included in the new **Connectors** PropertyManager.

Spring Connector

A spring connector connects a face of a component to a face of another component by distributed springs with specified normal and tangential stiffnesses. The two faces must be planar and parallel to each other. The springs are introduced in the common area of projection of one of the faces onto the other. The springs resist tension and compression.



Spring connectors are introduced in the common area of projection of the two faces.

If a distributed stiffness is specified, the equivalent total stiffness is equal to the specified value times the area of projection.

Tips

- More accurate results are obtained when each face is identical to the projection of the other face onto it. If the original faces do not satisfy this criterion, you can do one of the following things to improve the accuracy:

  Split one or both faces, as needed, by projecting one face onto the other and then define a spring between the aligned faces resulting from projection.

  Define two spring connectors with reverse selection order specifying half the stiffnesses for each connector.

- No springs are created if the faces have no common area of projection.
- The faces can be coinciding. This enables you to simulate the stiffness of a thin elastic layer of material separating the two faces without actually modeling the material or leaving a gap for the space it occupies. The imaginary springs in this case are initially of zero length.
- When visualizing results, it is important to plot the deformed shape with 1.0 scale factor to make sure that no interference between parts has developed. When interference occurs, the results are not valid. You can define contact conditions between the interfering faces to eliminate this unrealistic condition before re-running the study.

Pin Connector

A pin connector connects a cylindrical face of a component to a cylindrical face of another component. Each face must be a full cylinder (360^0). The two faces can have different radii but must have coinciding axes. Each face can be inner (hole) or outer (solid). The following restrictions are enforced in all cases:

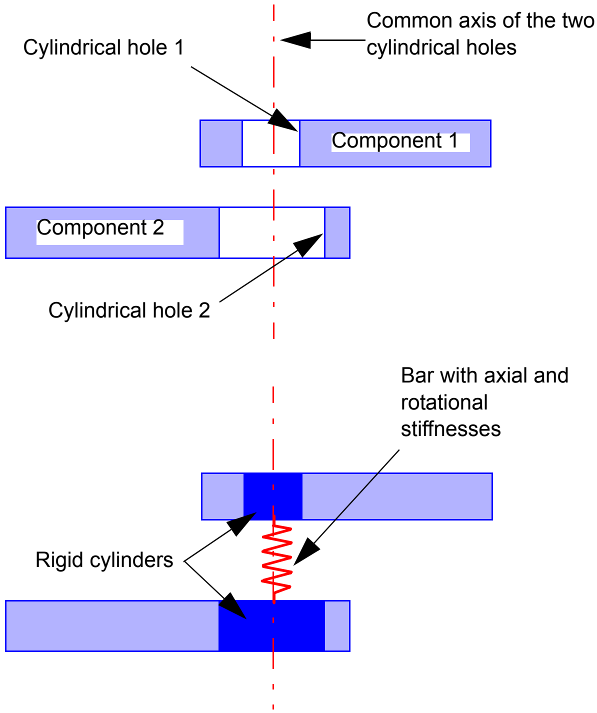
- The pin itself remains straight (it does not bend)
- Each face maintains its original shape but can move as a rigid body
- The two faces remain coaxial

The simulation is based on the small displacement assumption and it does not consider inertial effects. Inertial effects can be considered by importing motion loads from COSMOSMotion.

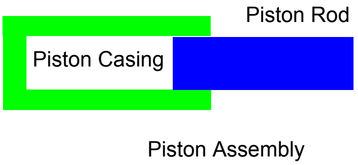
The program simulates the pin by making the cylindrical faces rigid and then connecting them by a bar with specified axial and rotational stiffnesses (springs). The relative axial movement of the faces depends on the axial force that develops in the joint and the specified axial stiffness. Similarly, the relative rotation is based on the moment that develops in the joint and the specified rotational stiffness.

The figures below illustrate some practical situations for using pins:

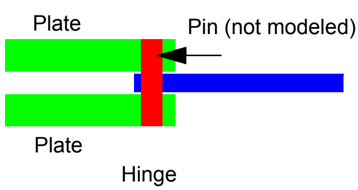
Connecting two cylindrical holes with a pin:



Connecting a cylindrical hole and a cylindrical solid:



Define a pin between the inner face of a piston casing and the face of a piston rod. Use an axial spring to specify the resistance to sliding of the rod into the casing. The pin geometry is created.



Define 2 pins for this joint. The actual stiffnesses are the summation of the corresponding individual values. Although the springs appear to be in series, they are treated as if they are connected in parallel. The total stiffness is the summation of the two stiffnesses. The pin geometry is not created.

A pin defined between the cylindrical faces will NOT WORK properly because the faces do not represent full cylinders.

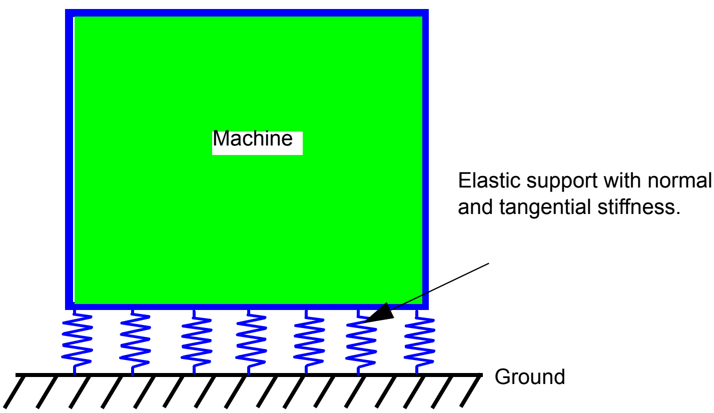
The behavior of the pin is controlled by its resistance to translation and rotation. If **No rotation** is checked, the pin prevents relative rotation between the two cylindrical faces. Similarly, if **No translation** is checked, the pin prevents relative axial translation between the two cylindrical faces. If **No rotation** is not checked, the program assumes a smooth hinge (zero stiffness) or you can specify a value for **Rotational stiffness**. Similarly, if **No translation** is not checked, the program assumes there is no resistance to relative axial translation (zero stiffness) or you can specify a value for **Axial stiffness**.

Tips

- If faces involved in the pin connection are initially touching, you must use the **Free** contact condition. Otherwise they are bonded by the mesher and the pin will not work.
- Due to introducing rigid regions, the stresses near the area of the pin may not be accurate. The effect decreases gradually until it practically disappears in regions about 1 diameter away from the cylindrical faces. For more accurate modeling of the pin, you need to create the pin and specify proper contact conditions.
- When viewing the results, it is important to plot the deformed shape with 1.0 scale factor to make sure that no interference between parts has developed due to the pin connection. If interference is present, the results are NOT valid. You can specify contact conditions between the interfering faces to eliminate this unrealistic condition before remeshing and running the study.

Elastic Support Connector

An elastic support connector connects the selected faces of a part or assembly component to the ground by specified normal and tangential stiffnesses. The faces do not have to be planar. A distributed stiffness at a point on the face represents the stiffness density associated with an infinitely small area around that location. The tangential stiffness at a location is assumed to be equal in all directions tangential to the face at that point. Use this option to simulate elastic foundations of machines and other objects. The elastic support can be made of actual springs or a layer of an elastic material.



Summary of Connectors

The following table summarizes options available for defining connectors.

Connector Type	**Geometric Entities**	**Reference Geometry Type**	**Required Input**
Rigid	Faces of solids from different assembly components.	N/A	N/A
Pin	Two Coaxial Cylindrical Faces from different assembly components.	N/A	Axial stiffness if the pin is not free to translate and tangential stiffness if the pin is not free to rotate.
Spring	Two Planar and Parallel Faces from different assembly components.	N/A	Axial and tangential stiffnesses.
Elastic Support	Faces of solids of a part or assembly.	N/A	Axial and tangential stiffnesses.

Thermal Loads and Restraints

Except for temperature, which is common to thermal and structural studies, the other thermal loads and restraints are only accessible for thermal studies.

For steady state thermal studies with a heat source, a mechanism for heat dissipation must be defined. Otherwise, analysis stops because the temperatures increase without bound. Transient thermal studies run for a relatively short period of time and thus do not require a heat dissipation mechanism.

The following types of loads and restraints are available for thermal studies:

- **Temperature** (prescribed temperatures)
- **Convection**
- **Heat Flux**
- **Heat Power**
- **Radiation**

Temperature

The temperature of an entity with a prescribed temperature remains constant at all times. Depending on other thermal loads and restraints, the model can lose or gain thermal energy at locations with prescribed temperatures. Prescribed temperatures can be applied to vertices, edges, faces, and components.

Convection

Convection describes the heat transfer mechanism between a solid face and an adjacent moving fluid (or gas). It involves the combined effects of conduction and the moving fluid. Fluid particles act as carriers of thermal energy.

The rate of heat exchange between a fluid of a bulk temperature $\mathbf{T_f}$ and a face of a solid of area **A** and temperature $\mathbf{T_s}$ is expressed as:

$$\dot{Q}_{convection} = hA(T_s - T_f)$$

where **h** is the convection heat transfer coefficient.

A solid loses thermal energy through a face with convection if the temperature of the face is higher than the bulk temperature. The solid gains thermal energy if the temperature of the face is lower than the bulk temperature.

Radiation

The radiation rate of heat transfer between a face of area **A** at temperature $\mathbf{T_s}$ **and the ambient** is expressed as:

$$\dot{Q}_{radiation} = f\varepsilon\sigma A(T_s^4 - T_a^4)$$

Where f is the view factor, ε is the emissivity, and σ is the Stefan-Boltzmann constant.

You must define the emissivity (ε), the ambient temperature (T_a), and the view factor (f) to define radiation.

Heat Flux

Heat flux applied to a face specifies the rate of thermal energy transfer per unit area of the face.

Heat Power

Heat power specifies the rate of thermal energy generated at vertex, edge, or face. If you select multiple entities, the program applies the specified value to EACH entity.

Free Faces

Free faces (faces without any boundary conditions) are insulated. A free face is thermally similar to a face with a zero temperature gradient in the normal direction. Faces of thermal symmetry can be modeled as free faces. Heat can flow parallel to the face but cannot flow normal to the face.

Summary of Thermal Loads and Restraints

The following table summarizes options available for thermal loads and restraints.:

Load Type	Geometric Entities	Reference Geometry Type	Required Input
Temperature	Vertices, Edges, Faces, and components	N/A	Unit and temperature value.
Convection	Faces	N/A	Film coefficient and bulk temperature in the desired units.

Load Type	Geometric Entities	Reference Geometry Type	Required Input
Radiation	Faces	N/A	Unit and value of surrounding temperature, emissivity, and view factor
Heat Flux	Faces	N/A	Unit and value of the heat flux (heat power/unit area). Temperature range for optional thermostat for transient studies.
Heat Power	Vertices, Edges, Faces, components, and an optional vertex for thermostat location for transient studies	N/A	Unit and value of the heat power. The specified value is applied to *each* selected entity. Temperature range for optional thermostat for transient studies.

COSMOSWorks supports thermal contact resistivity between pairs of faces in an assembly. Refer to the *Meshing* chapter for more information and to the COSMOSWorks Online Tutor for an example.

Multiple Application of Thermal Loads

Thermal loads and boundary conditions overwrite each other. Newer assignments of temperature, convection, radiation, heat flux, and heat power overwrite previous assignments. The latest assignment for each entity is used.

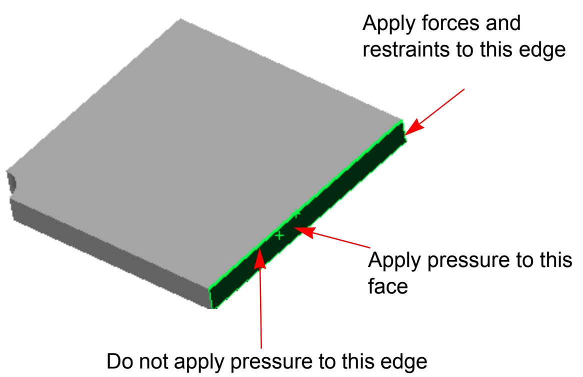
Applying Loads and Restraints to Shells

The way you apply loads and restraints to shell models depends on how the shell model was created. When defining a study, two options are available for the mesh type: **Shells using midsurfaces** (for part documents only) and **Shell using surfaces**. Here are some tips that should be considered when applying loads and restraints for such models.

Shell using midsurfaces

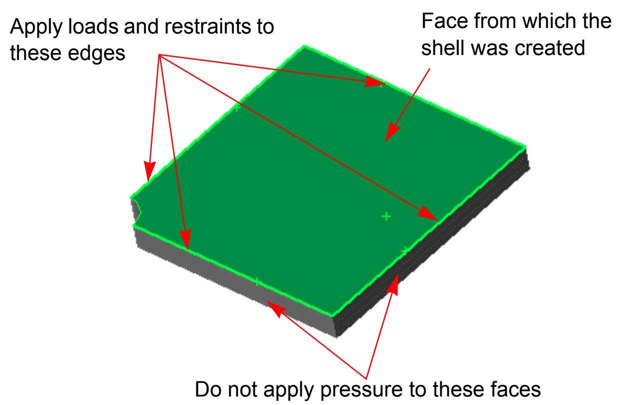
- To apply forces or restraints to a shell vertex, select the associated edge of the solid.
- To apply pressure to a shell edge, select the associated face of the solid.

Since shell elements have rotational degrees of freedom, you must differentiate between the **Immovable** restraint (no translation) and the **Fixed** restraint (no translation and no rotation). You can also apply concentrated moments using the **Force** PropertyManager.



Shell using surfaces

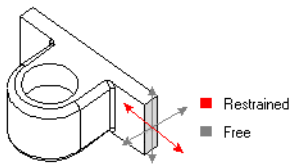
- Apply loads and restraints to the edges and vertices of the surfaces (or faces of solids) you used to create the shells.
- When applying pressure to a shell edge, the pressure is specified per unit area. The program internally uses the thickness of the shell. The equivalent force applied to the edge is equal to the pressure value times the length of the edge times the thickness of the shell.



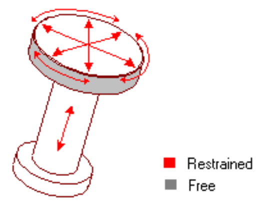
Applying pressure to edges of a shell model created using the top face of the plate

Miscellaneous Examples

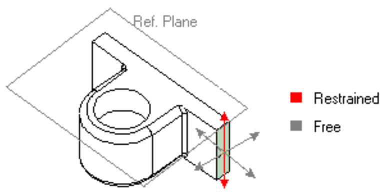
The following figures provide examples of applying loads and restraints.



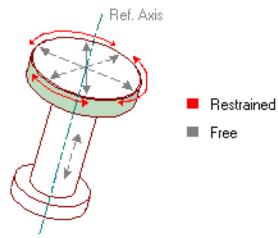
Restrained Normal to the Face



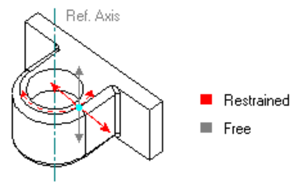
Cylindrical Face



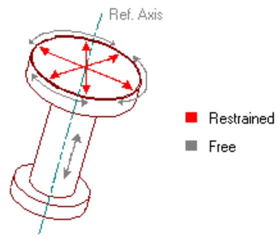
Restrained Normal to Ref. Plane



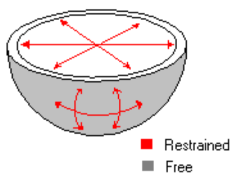
Circumferentially Restrained to Ref. Axis



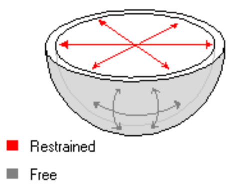
Radially and Circumferentially Restrained to Ref. Axis



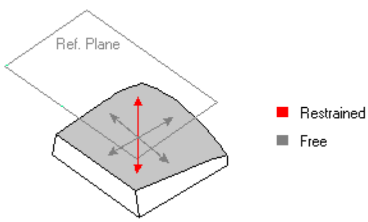
Edge Radially Restrained to Ref. Axis



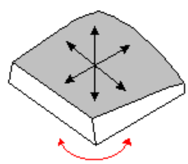
Spherical Face



Radially Restrained Spherical Face



Lofted Face Restrained Normal to Ref. Plane



Fixed Arbitrary Face

6

Meshing

Meshing is a crucial step in performing analysis to your model. This chapter discusses the following topics:

- **Solid Mesh**. Describes solid meshing and tetrahedral elements.
- **Shell Mesh**. Describes shell meshing and shell elements.
- **Shell Modeling**. Describes steps to perform shell modeling.
- **Meshing Parameters**. Describes parameters affecting the generation of a mesh.
- **Contact Options for Static and Thermal Studies**. Learn about contact functionality and how and when to use different contact options.
- **Mesh Quality Check**. Explores checking performed by the program to check the quality of the generated elements in a mesh.
- **Mesh Failure Diagnostics**. Learn about this functionality and how to use it efficiently when meshing assemblies.

Background

Finite Element Analysis (FEA) provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes (elements) connected at common points (nodes). Finite element analysis programs look at the model as a network of discrete interconnected elements.

The Finite Element Method (FEM) predicts the behavior of the model by manipulating the information obtained from all the elements making up the model.

Meshing is a very crucial step in design analysis. The automatic mesher in COSMOSWorks generates a mesh based on a global element size, tolerance, and local mesh control specifications. Mesh control lets you specify different sizes of elements for components, faces, edges, and vertices.

COSMOSWorks estimates a global element size for the model taking into consideration its volume, surface area, and other geometric details. The size of the generated mesh (number of nodes and elements) depends on the geometry and dimensions of the model, element size, mesh tolerance, mesh control, and contact specifications. In the early stages of design analysis where approximate results may suffice, you can specify a larger element size for a faster solution. For a more accurate solution, a smaller element size may be required.

Meshing can generate 3D tetrahedral solid elements or 2D triangular shell elements depending on the selected option when the study is created. Shell elements are naturally suitable for modeling thin parts (sheet metals). Using tetrahedral (solid) elements in meshing thin parts can be very inefficient due to the large number of elements that can be generated.

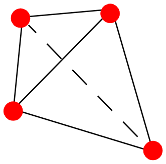
Solid Mesh

In meshing a part or an assembly with solid elements, COSMOSWorks generates one of the following types of elements based on mesh preferences:

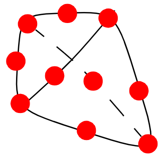
- **Draft quality mesh**. The automatic mesher generates linear tetrahedral solid elements.
- **High quality mesh**. The automatic mesher generates parabolic tetrahedral solid elements.

Linear elements are also called first-order, or lower-order elements. Parabolic elements are also called second-order, or higher-order elements.

A linear tetrahedral element is defined by four corner nodes connected by six straight edges. A parabolic tetrahedral element is defined by four corner nodes, six mid-side nodes, and six edges. The following figures show schematic drawings of linear and parabolic tetrahedral solid elements.



Linear tetrahedral element

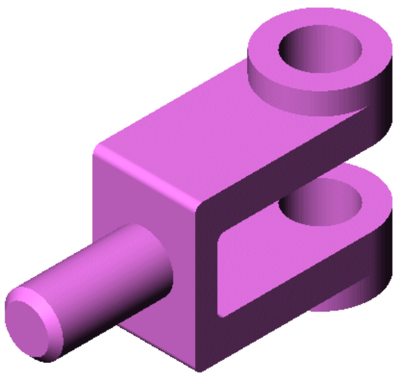


Parabolic tetrahedral element

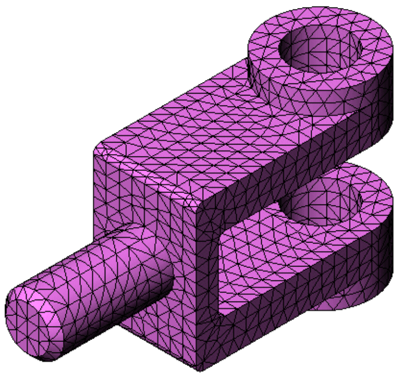
For the same mesh density (number of elements), parabolic elements yield better results than linear elements because: 1) they represent curved boundaries more accurately, and 2) they produce better mathematical approximations. However, parabolic elements require greater computational resources than linear elements.

Each node in a solid element has three degrees of freedom which are the translations in three orthogonal directions. COSMOSWorks uses the X, Y, and Z directions of the global Cartesian coordinate system in formulating the problem.

For thermal problems, each node has one degree of freedom which is the temperature.



CAD model of a part



Meshed model

High quality mesh is recommended for final analysis.

Shell Mesh

When using shell elements, COSMOSWorks generates one of the following types of elements depending on the active settings in the **Mesh** preferences dialog box:

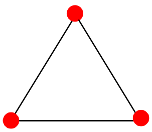
- **Draft quality mesh**. The automatic mesher generates linear triangular shell elements.

- **High quality mesh**. The automatic mesher generates parabolic triangular shell elements.

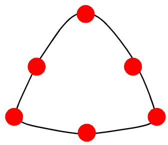
A linear triangular shell element is defined by three corner nodes connected by three straight edges. A parabolic triangular element is defined by three corner nodes, three mid-side nodes, and three parabolic edges. For studies created with the Shell using Midsurface option, the thickness of the elements is automatically extracted from the geometry of the model.

To set the desired option, right-click the **Mesh** folder and select **Preferences**.

Shell elements are 2D elements capable of resisting membrane and bending loads.



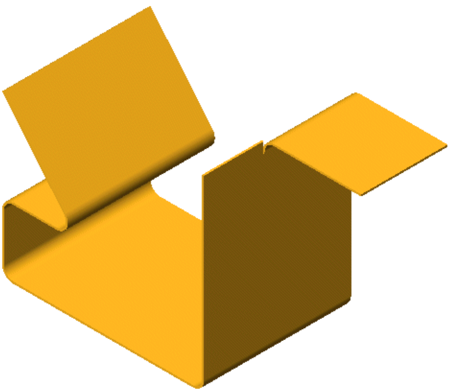
Linear triangular shell element



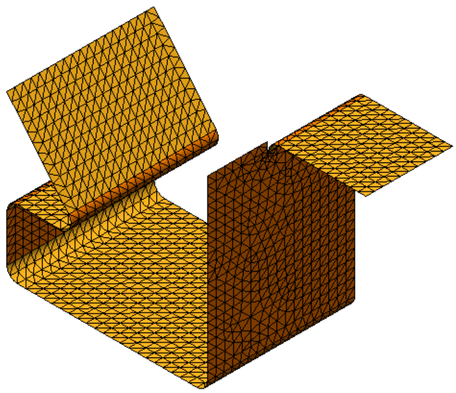
Parabolic triangular shell element

For structural studies, each node in shell elements has six degrees of freedom; three translations and three rotations. The translational degrees of freedom are motions in the global X, Y, and Z directions. The rotational degrees of freedom are rotations about the global X, Y, and Z axes.

For thermal problems, each node has one degree of freedom which is the temperature.



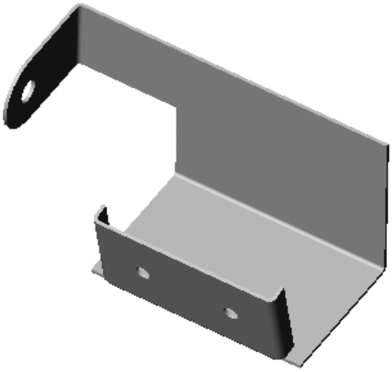
Sheet metal part



Shell model

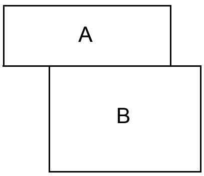
A reasonably fine draft quality mesh gives results that are generally similar to results obtained from a high quality mesh with the same number of elements. The difference between the two results increases if the model includes curved geometry.

Shell Modeling

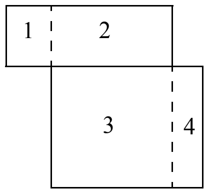


You can mesh any solid model with tetrahedral solid elements. However, meshing thin models with solid elements results in generating a large number of elements since you have to use a small element size. Using a larger element size deteriorates the quality of the mesh and leads to inaccurate results. Shell meshing is the natural choice for sheet metal and thin parts. Surface models can only be meshed with shell elements.

For proper shell meshing of surface models, faces should share full edges. Otherwise, the model may not be compatible along the interface. Split surfaces to maintain this condition.



Surfaces A and B do not share full edges. Mesh may or may not be compatible.



Splitting surfaces A and B as shown generates two additional faces. After splitting, all four surfaces share full edges. The mesh will be compatible.

Surface knitting can help resolve some of edge incompatibility situations automatically. To access surface knitting, click **Insert**, **Surface**, **Knit**.

How to Model Shell Problems

The main steps in shell meshing are as follows:

Defining a Study

This is where you decide whether you will use solid or shell meshing. There are two options for shell modeling:

Shell mesh using mid-surfaces

When selecting this option, the program extracts mid-surfaces automatically. Only one material can be assigned to the part. This option is not available for assemblies. COSMOSWorks extracts the mid-surfaces and assigns thicknesses automatically. The mid-surface of a generated shell element coincides with the associated extracted mid-surface. Use this option for sheet metals and simple thin parts.

Shell mesh using surfaces

This option lets you select the faces to be meshed and allows you to assign the desired thicknesses and materials. You can repeat this process as many times as desired. Adjacent shells are bonded automatically. Since this option works on arbitrary faces and surface models, it is supported for parts and assemblies. The mid-surface of a generated shell element coincides with the associated face or surface.

To create shells, select the desired face(s) or surface(s), and select **Define By Selected Surfaces**. To create a shell for every face or surface in the model automatically, right-click the **Shells** folder and select **Define by All Ref Surfaces**.

Assigning Materials

For shell using mid-surfaces

Only one material is allowed for a shell model created using the **Shell mesh using mid-surfaces** option. One icon is created in the **Mid-surface Shell** folder.

For shell using surfaces

Assigning a material for a shell model created using the **Shell mesh using surfaces** option is similar to assigning material for a solid component in an assembly. The program creates a **Shells** folder. An icon is created in the **Shells** folder for every face selected for shell modeling. You can assign a different material to every item in the **Shells** folder, just like the case with the **Solids** folder of an assembly.

Assigning Thickness

For shell using mid-surfaces models

The program automatically extracts and assigns a uniform thickness for each pair of faces. Thin shell formulation is used.

For shell using surfaces

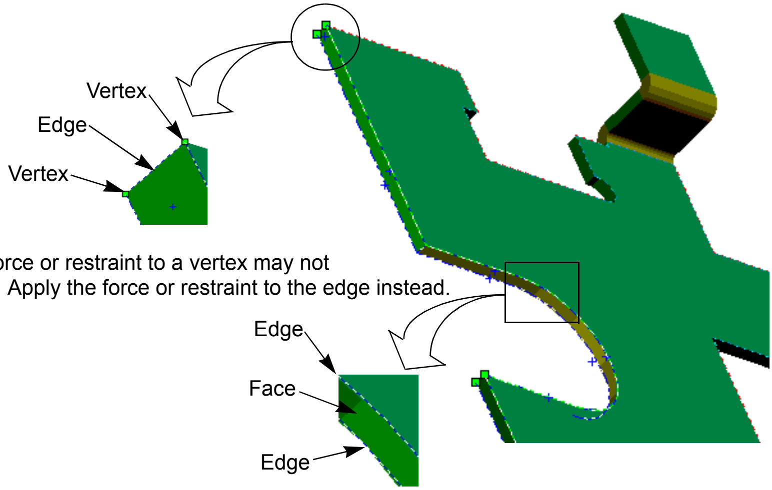
Whenever you create a shell on a face or surface, a PropertyManager appears to let you assign a thickness value and select thin or thick shell formulation. An icon is created in the **Shells** folder for each shell. Each shell can have a different thickness, material, and thin or thick formulation.

As a general guideline, thin shells can be used when the thickness to span ratio is less than 0.05.

Applying Restraints, Loads, and Mesh Control

For shell using mid-surfaces

- To apply a restraint or a load to a shell edge, select the associated face of the solid.
- To apply a restraint, load, or mesh control to shell vertex, select the associated edge of the solid.



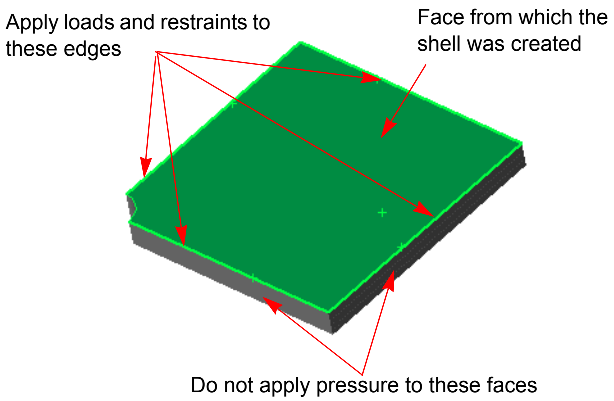
Applying a force or restraint to a vertex may not always work. Apply the force or restraint to the edge instead.

Applying a force or restraint to an edge may not always work. Apply the force or load to the face instead.

Since shell elements have rotational degrees of freedom, you must differentiate between the **Immovable** restraint (no translation) and the **Fixed** restraint (no translation and no rotation). You can also apply concentrated moments using the **Force** PropertyManager.

For shell using surfaces or faces of Solids

- Apply load or restraint to the edges and/or vertices of the surfaces (or faces of solids) you used to create the shells.
- You can apply pressure to a shell edge. The pressure is specified per unit area. The program internally uses the thickness of the shell. The equivalent force applied to the edge is equal to the pressure value times the length of the edge times the thickness of the shell.



Applying pressure to edges of a shell model created using the top face of the plate

Meshing

Apply the desired mesh control on the appropriate faces, edges, and vertices. Before meshing, verify the active meshing preferences, and specify all desired mesh controls. To generate a shell mesh, right-click the **Mesh** icon and select **Create**.

When creating a shell, you can select thin or thick shell formulation. Thin shells can be used when the length to span ratio is equal to or less than 0.05. You may not mix shell and solid elements in this release.

Flipping Shell Elements

Shell elements are created at the mid-surface. A shell element has a bottom face and a top face. Stress results on top and bottom faces of shell elements are generally different. Since stresses at a node are calculated by averaging nodal results from all elements meeting at that node, it is essential that the elements have the proper orientation. When elements are oriented properly, bottom stresses at a node common to several elements are calculated by averaging bottom stresses of the associated elements. If shell elements are not oriented properly, bottom and top stresses are mixed up leading to incorrect stress plots. It is important, therefore, to check the orientation of shell elements right after generating the mesh and before running the study.

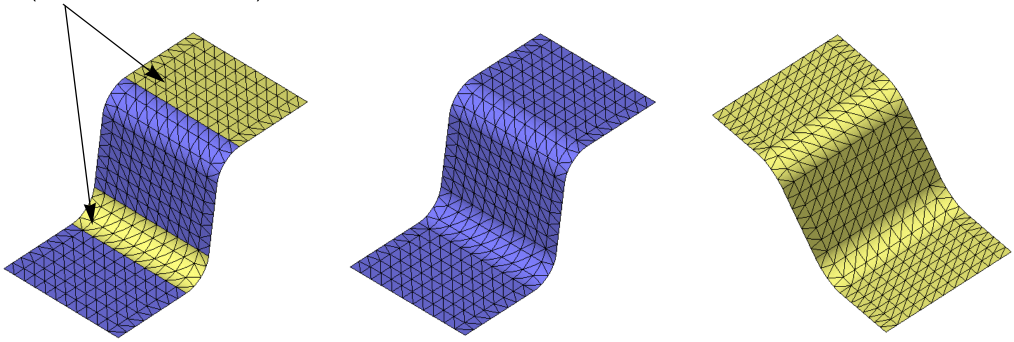
To check the orientation of the shell elements, show the mesh in a shaded view. The bottom faces are displayed in the color selected for **Shell bottom face color** in the **Mesh** preferences dialog box.

To flip shell elements on a face:

1 Display the shell mesh and identify the faces of the model whose associated shell elements need to be flipped.

2 In the graphics area, click the face(s) to be flipped.

3 In the COSMOSWorks Manager tree, right-click the **Mesh** icon and select **Flip shell elements**.



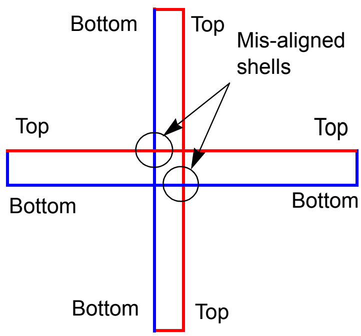
Elements on the upper faces are not oriented properly

Elements on the upper faces are flipped to have the proper orientation

Bottom faces of all shell elements have same color

T-Shaped and Intersecting Shells

While it is possible to orient shells on sheet metals properly, it is not possible to do so for T-shaped and intersecting shells. In such cases, some edges will have mis-aligned elements. As a result, nodal stresses are not correct along these edges. We recommend to display elemental stresses for such models since no cross-element averaging is involved.



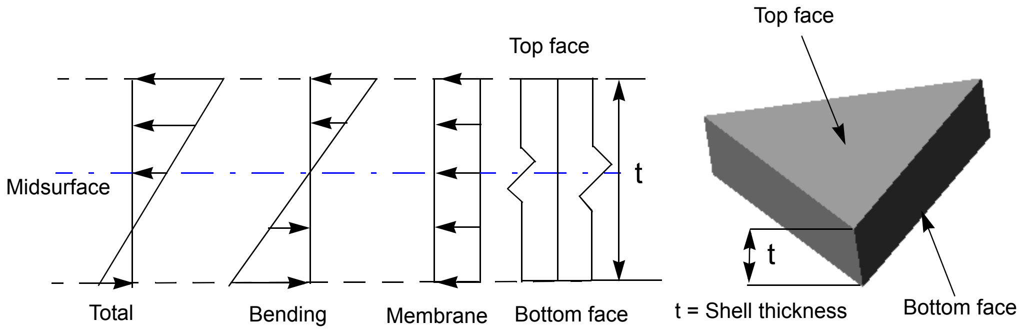
Viewing of Stress Results for Shells

The dialog boxes for plotting and listing stress results let you select one of the following options:

- **Top** (bending + membrane stresses on the top face)
- **Bottom** (bending + membrane stresses on the bottom face)
- **Membrane** (membrane stress component)
- **Bending** (Bending stress component)

These options are illustrated in the following figure:

Refer to the *Viewing Results* chapter for more details.



Identifying Faces of Shells

To identify the top and bottom faces of a shell mesh:

In the COSMOSWorks Manager tree, right-click the **Mesh** icon and select **Show Mesh**.

Top faces of the shell appear in the color of the model in the shaded view mode. The bottom faces appear in the **Shell bottom face color** specified in the **Mesh** preferences dialog box.

> Although displacement results in static studies and temperature results in thermal studies do not change if the shells are not aligned, it is recommended to align the shells for all studies.

Rebuilding the Mesh

You can use one mesh for multiple studies to investigate the effect of using different materials, loads, and restraints. However, to consider the impact of geometry changes on the results, you must rebuild the mesh and rerun the study after making any change in geometry. The old mesh will be overwritten, but you can still view the mesh associated with a study and view the associated results. If you rerun a study after rebuilding the mesh, the new mesh will be used and all old results will be overwritten.

You can create solid and shell studies in the same document.

To run a study using its associated (old) mesh, activate the study by clicking on its icon, show its mesh by right-clicking the **Mesh** icon and selecting **Show Mesh**, and then run the study.

Meshing Parameters

The mesh is generated by right-clicking the **Mesh** icon in the COSMOSWorks Manager tree and selecting **Create**. The generated mesh depends on the following factors:

- Mesh type (solid, shell mesh using mid-surfaces, or shell mesh using surfaces) of the active study
- Active mesh preferences
- Mesh control specifications
- Contact/Gaps options
- Global element size (specified in the Mesh PropertyManager)
- Tolerance (specified in the Mesh PropertyManager)

It is recommended that you verify all these factors before meshing. Any change in these factors requires remeshing. The **Preferences** button in the **Mesh** PropertyManager provides a convenient access to check meshing preferences. Right-click the **Mesh** icon and select **Details** to view how an existing mesh was generated.

Mesh Preferences

Meshing preferences are essential factors in determining the quality of the mesh and hence the results. Results based on different preference settings should converge to each other if an adequately small element size is used.

You can set the **Mesh quality** to **Draft** or **High**. A draft quality mesh does not have mid-side nodes. Draft quality can be used for quick evaluation and in solid models where bending effects are small. High quality mesh is recommended in most cases, especially for models with curved geometry.

The **Standard** mesher uses the *Voronoi-Delaunay* meshing scheme for subsequent meshing operations. This mesher is faster than the alternate mesher and should be used in most cases. Try the **Alternate** mesher only when the standard mesher keeps failing. The Alternate mesher ignores mesh control and automatic transition settings.

The **Jacobian check** sets the number of integration points to be used in checking the distortion level of high order tetrahedral elements.

Automatic transition automatically applies mesh controls to small features, details, holes, and fillets. Uncheck **Automatic transition** before meshing large models with many small features and details to avoid generating a very large number of elements unnecessarily.

The **Smooth surface** option, when checked, results in slightly relocating the boundary nodes to improve the mesh. It is recommended to check this option in most cases.

Automatic looping instructs the mesher to automatically retry to mesh the model using a smaller global element size. You control the maximum number of trials allowed and the ratio by which the global element size and tolerance are reduced each time.

Setting the color for plotting the bottom faces of shell elements helps you align shell elements properly.

Mesh Control

Mesh control refers to specifying different element sizes at different regions in the model. A smaller element size in a region improves the accuracy of results in that region. You can specify mesh control at vertices, edges, faces, and components. To access the **Mesh Control** PropertyManager, right-click the **Mesh** icon and select **Apply Control**.

Mesh Control Parameters

Mesh control involves transition from one element size to another. The transition is defined by the following parameters:

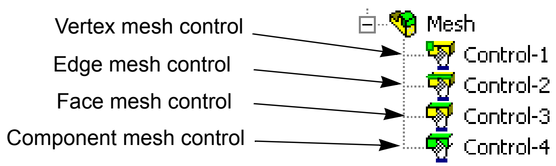
- **Element Size** . Element size for the specified entity (e)
- **Ratio** . Element growth ratio (r)
- **Layers** . Number of layers of elements (n)

Assuming that the global element size used for meshing is (E). The average element size in layers radiating from the entity will be:

Layer	Element Size
1	e
2	e*r
3	$e*r^2$
4	$e*r^3$
.	.
n	$e*r^n$

If the calculated average element size of a layer exceeds (E), the program uses (E) instead. If the specified number of layers (n) is too small for a smooth transition, the program adds more layers automatically.

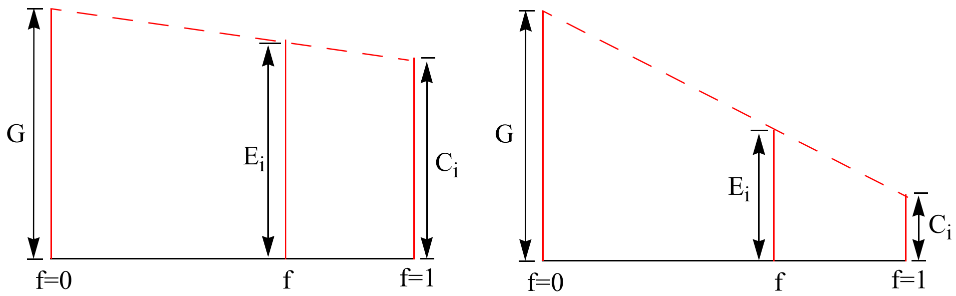
The mesh radiates from vertices to edges, from edges to faces, from faces to components, and from a component to connected components.



Component Mesh Control

When specifying component mesh control, you can specify a uniform element size for the selected components, or you can use a relative size that is interpreted based on individual components.

The relative size interpolates a different element size for each selected component based on the selected position on the slider. The left end of the slider corresponds to the default global element size of the assembly (G). The right end of the slider is different for different components. For each component, it corresponds to the default element size if the component is meshed independently as a part (C_i). Using C_i to mesh a component generates 4000 to 6000 elements in most cases.



Element size (E_i) used to mesh component (i) is smaller for smaller components

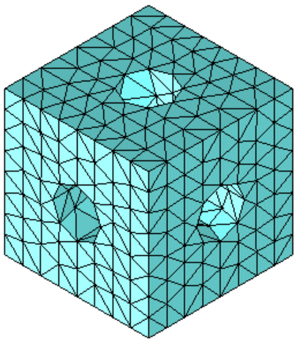
The program calculates the element size E_i for component i from the equation:

$$E_i = G - (G - C_i)f$$

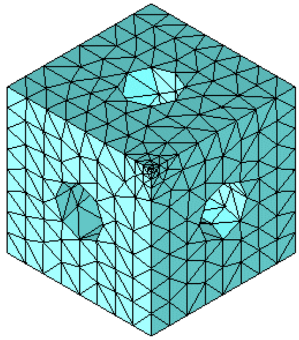
Where f refers to the selected position on the slider with f = 0 at the left end and 1.0 at the right end. This equation calculates smaller element sizes for smaller components as G is always greater than C_i.

Mesh Control Examples

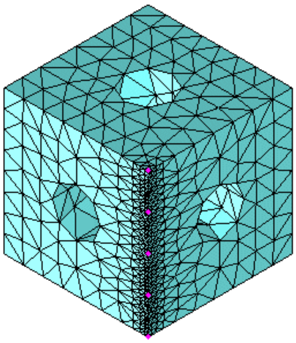
The following pictures illustrate mesh control on various entities.



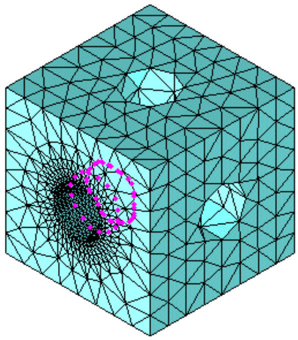
Uniform meshing (no mesh control applied)



Mesh control applied to a vertex



Mesh control applied to an edge



Mesh control applied to a face

Contact Options for Static and Thermal Studies

Contact options are used for static and thermal studies of assembly models only. The **Contact/ Gaps** icon appears above the **Mesh** icon in the COSMOSWorks Manager tree. The right-mouse menu for the **Contact/Gaps** folder contains the following options:

- Touching Faces: Bonded
- Touching Faces: Free
- Touching Faces: Node to Node

One of these options must be selected. We refer to the selected option as the *global* contact condition.

In addition, the menu contains two more options:

- Define Contact Pair (we refer to this as *local* or *face-to-face* contact)
- Define Contact for Components (we refer to this as the *component* contact)

The global contact setting applies to all touching faces, but you can define different face-to-face and component contact options.

Each component or face-to-face contact condition is represented by an icon in the **Contact/Gaps** folder.

> **Contact/Gaps** options are only supported for static and thermal analysis studies of assemblies using solid mesh. For thermal studies, use local surface contact to model thermal contact resistance.

Global Contact/Gaps Options

The following global options are available:

Touching Faces: Bonded

All touching faces of assembly components are bonded at their points of contact unless otherwise specified by local or component contact conditions. Components share nodes in the contact area. This is the default option.

Touching Faces: Free

All touching faces are free to move in any direction. The program allows components with free faces to move into each other under the effect of loads. This is a physical impossibility. Specifying this option for some faces can accelerate the contact iteration process. You should only use this option when you are absolutely sure that the specified loading will not cause the touching faces to penetrate each other.

Touching Faces: Node to Node

With this option, the program automatically creates compatible part meshes on touching part areas. The program then creates gap elements connecting corresponding nodes across the faces. A gap element between two nodes prevents them from moving closer to each other but allows them to move away from each other. This option can be used with the small displacement contact only. Large displacement contact does not support this option.

> Global specifications are used for partially and fully touching faces in the model for which no local or component settings have been defined.

Component Contact Options

You can apply contact conditions to the interface of selected components with other components. This saves you time because you do not have to specify contact conditions for pairs of faces individually. Component contact settings override the global contact settings.

Local Contact Options (Face-to-Face)

Local contact defines the type of contact for a pair of faces. In addition to the bonded, free, and node-to-node options, the following options are available for local contact: Local contact settings override the global and component contact settings.

When defining a local node-to-node or bonded contact conditions, the two faces must be initially touching each other fully or partially.

Surface

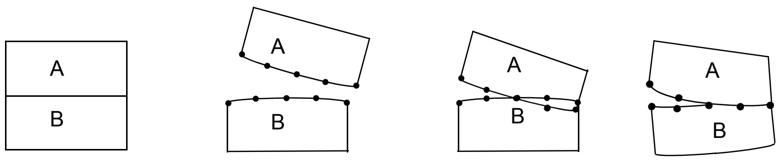
The program creates node-to-area pairs automatically. Each node from one of the faces is associated with an area defined by nodes on the other face. The faces can move away from each other but cannot penetrate each other. Infinitesimal sliding is considered. This option is not available for global or component contact. It can be used with the small as well as the large displacement contact options.

The surface contact condition allows you to specify thermal contact resistance for thermal studies. Refer to the *Thermal Contact Resistance* section for details.

Shrink Fit

The program creates a shrink fit condition between the selected faces. The faces do not have to be cylindrical.

The various contact options are illustrated in the following figures. Refer to the *Shrink Fit* section for details.



(a) Bonded: faces are bonded together at all times.

(b) Surface

(c) Free: Use this option only if you are sure that the two faces will not penetrate each other.

(d) Node to Node: Use this option only when the two faces have identical meshes. This condition is automatically met for mated faces. If you specify mesh control on one face, specify the same control on the other face.

Multiple Contact Conditions

When multiple contact assignments are specified for an entity, they are enforced as follows:

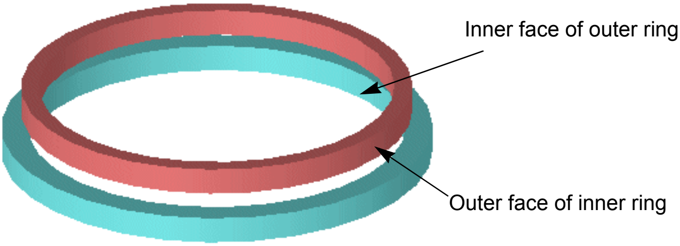
- Local (face-to-face) settings override global and component settings
- Component contact settings override global settings.
- The global contact condition will be used for all touching faces for which no component or local contact condition has been specified.

Shrink Fit

Shrink fit is encountered in many engineering designs. It refers to fitting a body into a slightly smaller cavity. Due to normal forces that develop at the interface, the inner part will shrink while the outer part will expand. The amount of shrinkage/expansion is determined by the material properties, geometry, loads, and restraints. Shrink fit is specified between a pair of faces.

The following figure illustrates a shrink fit contact condition between two rings. The outer diameter of the inner ring is slightly larger than the inner diameter of the outer ring. The contact condition is defined between the outer face of the inner ring and the inner face of the outer ring.

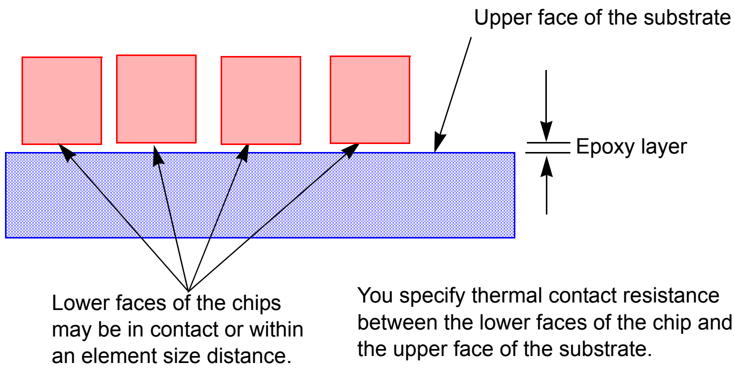
For accurate results, the overlap should be large enough to overcome approximations introduced by meshing. For example, the overlap of cylindrical or spherical faces should be larger than 0.1% of the diameter for accurate results.



In modeling shrink fit problems, the geometry should be modeled as is (with overlap). You must mesh after defining the shrink fit condition.

Thermal Contact Resistance

In the electronic industry, chips are usually joined to substrates by a thin layer of epoxy. Similar situations are encountered in other industries. Modeling the epoxy layer as a separate component requires the use of a very small element size that can result in meshing failure or an unnecessarily large number of elements.

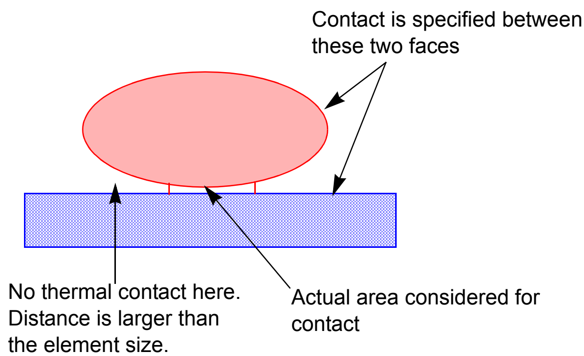


To consider the thermal resistance caused by the epoxy layer, you do not need to model it. Thermal contact resistance is implemented as a surface-to-surface contact condition. You can either specify the total resistivity or the resistivity per unit area.

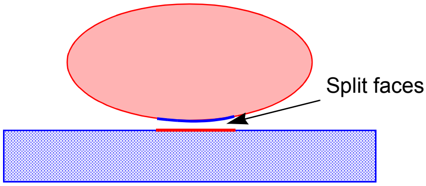
Modeling Thermal Contact Resistance

There are two ways of modeling thermal contact resistance:

1. You can neglect the thin layer of epoxy when creating the geometry. In other words, the faces of the components that are separated by the thin layer in reality, will be touching in the model.
2. You can consider the thin epoxy layer when creating the geometry. In this case there will be a gap between the faces of thermal contact. When using this approach, there are two points to consider:
   - Thermal contact is considered only if the gap is less than or equal to the element size in the neighborhood. As an example, if thermal contact is specified between the two larger faces shown in the figure, it is only considered for portions of the faces that are within one element size from each other.

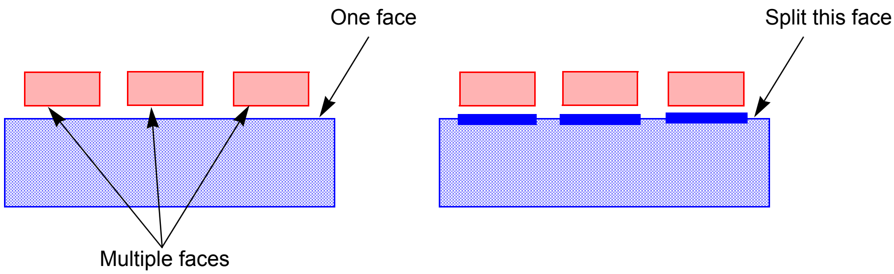


- Splitting the faces for proper pairing of thermal contact, although not necessary, improves the accuracy.



Splitting faces results
in better accuracy

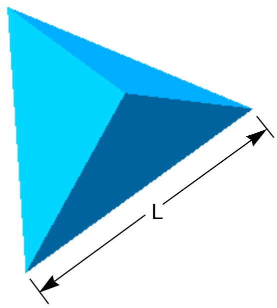
- To specify different thermal resistances between a large face and a number of smaller faces, you must first split the large face to a number of smaller faces before assigning thermal contact resistance for different pairs.



Modeling thermal resistance

The Mesh PropertyManager

To access the **Mesh** PropertyManager, right-click the **Mesh** icon of the desired study and select **Create**, or click **COSMOSWorks**, **Mesh**.



L = Global element size

The slide bar lets you change the global element size and tolerance. The extreme left position (Coarse) sets the global element size to twice the default size. The extreme right position (Fine) sets the global element size to half the default size. You can type in the desired value in the field.

The default global size suggests a value based on the volume and surface area of the model. The default tolerance is 5% of the global element size. Adjusting the tolerance can help resolve some meshing problems. For example, if meshing fails due to free edges, increasing the tolerance can solve the problem. The tolerance cannot exceed 30% of the element size.

Mesh Quality Check

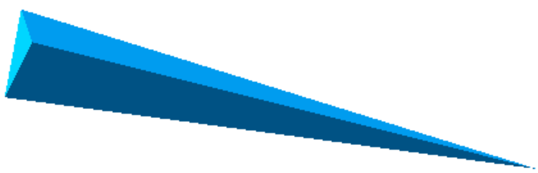
The quality of the mesh plays a key role in the accuracy of the results. COSMOSWorks uses two important checks to measure the quality of elements:

- Aspect ratio check
- Jacobian check

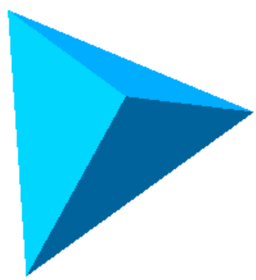
Aspect Ratio Check

For a solid mesh, numerical accuracy is best achieved by a mesh with uniform perfect tetrahedral elements whose edges are equal in length. For a general geometry, it is not possible to create a mesh of perfect tetrahedral elements. Due to small edges, curved geometry, thin features, and sharp corners, some of the generated elements can have some of their edges much longer than others. When the edges of an element become much different in length, the accuracy of the results deteriorates.

The aspect ratio of a perfect tetrahedral element is used as the basis for calculating aspect ratios of other elements. The aspect ratio of an element is defined as the ratio between the longest edge and the shortest normal dropped from a vertex to the opposite face normalized with respect to a perfect tetrahedral. By definition, the aspect ratio of a perfect tetrahedral element is 1.0. The aspect ratio check assumes straight edges connecting the four corner nodes. The aspect ratio check is automatically used by the program to check the quality of the mesh.



Tetrahedral element with relatively high aspect ratio



Tetrahedral element with aspect ratio of 1.0

Jacobian Check

Parabolic elements can map curved geometry much more accurately than linear elements of the same size. The mid-side nodes of the boundary edges of an element are placed on the actual geometry of the model. In extremely sharp or curved boundaries, placing the mid-side nodes on the actual geometry can result in generating distorted elements with edges crossing over each other. The Jacobian of an extremely distorted element becomes negative. An element with a negative Jacobian causes the analysis program to stop.

The Jacobian check is based on a number of points located within each element. COSMOSWorks gives you a choice to base the Jacobian check on **4**, **16**, **29** points or **At Nodes**.

It is recommended to use the **At Nodes** option when using the p-method to solve static problems.

The Jacobian ratio of a parabolic tetrahedral element, with all mid-side nodes located exactly at the middle of the straight edges, is 1.0. The Jacobian ratio increases as the curvatures of the edges increase. The Jacobian ratio at a point inside the element provides a measure of the degree of distortion of the element at that location. COSMOSWorks calculates the Jacobian ratio at the selected number of Gaussian points for each tetrahedral element. Based on stochastic studies it is generally seen that a Jacobian Ratio of forty or less is acceptable. COSMOSWorks adjusts the locations of the mid-side nodes of distorted elements automatically to make sure that all elements pass the Jacobian check.

To set the number of integration points to be used in the Jacobian check, right-click the **Mesh** icon and select **Preferences**, then select the desired number of points under **Jacobian Check** and click **OK**.

Mesh Failure Diagnostics

When meshing fails, COSMOSWorks gives a message and stops unless the automatic mesh looping is active. A failure diagnostics tool is provided to help you locate and resolve solid meshing problems.

The meshing of a solid component consists of two basic phases. In the first phase, the mesher places nodes on the boundary. This phase is called surface meshing. If the first phase is successful, the mesher starts the second phase where it creates nodes in the inside, fills the volume with tetrahedral elements, and places mid-side nodes. Mid-side nodes are placed only if **High** is selected in the **Mesh quality** box in **Mesh** preferences dialog box.

Failure can occur during one of the two phases. After meshing has failed, right-click the **Mesh** icon and select **Failure Diagnostics**. The **Failure Diagnostics** PropertyManager lists and highlights the components that failed. For each component, it lists and highlights the faces and edges that caused the failure.

To identify the problem with a component, face, or edge, select it in the list box. The **Status** box lists the phase in which meshing failed and offers possible solutions.

The **Failure Diagnostics** PropertyManager is not available for shell models in this release.

Meshing Tips

- It is important to select the proper mesh type when creating a study. Use **Solid mesh** for bulky solid parts and assemblies. Use **Shell mesh using mid-surface** for sheet metals. If possible, select **Shell mesh using surfaces** for surface models, or complex thin solid models when the shell mesh using mid-surface option fails.
- For assemblies, check component interference. COSMOSWorks allows interference only when the shrink fit contact option is used.
- If meshing fails, use the **Failure Diagnostics** tool to locate the cause of mesh failure. Try the proposed options to solve the problem.
- It is good practice to check mesh preferences before meshing. For example, the **Automatic transition** can result in generating an unnecessarily large number of elements for models with many small features. The **Automatic looping** can help solve meshing problems automatically, but you can adjust its settings for a particular model. Try the **Alternate** mesher only when the **Standard** mesher keeps failing.
- To improve results in important areas, use mesh control to set a smaller element size. When meshing an assembly with a wide range of component sizes, default meshing results in a relatively coarse mesh for small components. Component mesh control offers an easy way to give more importance to the selected small components. Use this option to identify important small components.
- When using mesh control, verify the status of the mesh control specifications. Suppressed mesh control specifications are ignored by the mesher.

7

Design Optimization

This chapter introduces design optimization and explains how to perform optimization studies. It includes the following topics:

- **Product Development Cycles**. Describes steps involved in product development.
- **Searching for the Optimum Solution**. Shows how the optimization algorithm works.
- **Using Optimization Studies**. Learn basic steps needed to define and run optimization studies.

Product Development Cycles

The design process is an iterative process in which a design is continuously modified until it meets acceptance criteria defined by safety, cost, performance, convenience, and shape. An initial design can go through many product development cycles before mass production. A product development cycle includes the following steps:

- Building the model in SolidWorks CAD system
- Prototyping the design
- Testing the prototype in the field
- Evaluating results from field tests
- Modifying the design based on the field test results

This process continues until a satisfactory solution is reached.

Design cycles are expensive and time-consuming due to prototyping and field testing. Because of time and cost constraints, most designers accept a solution that may not be optimum.

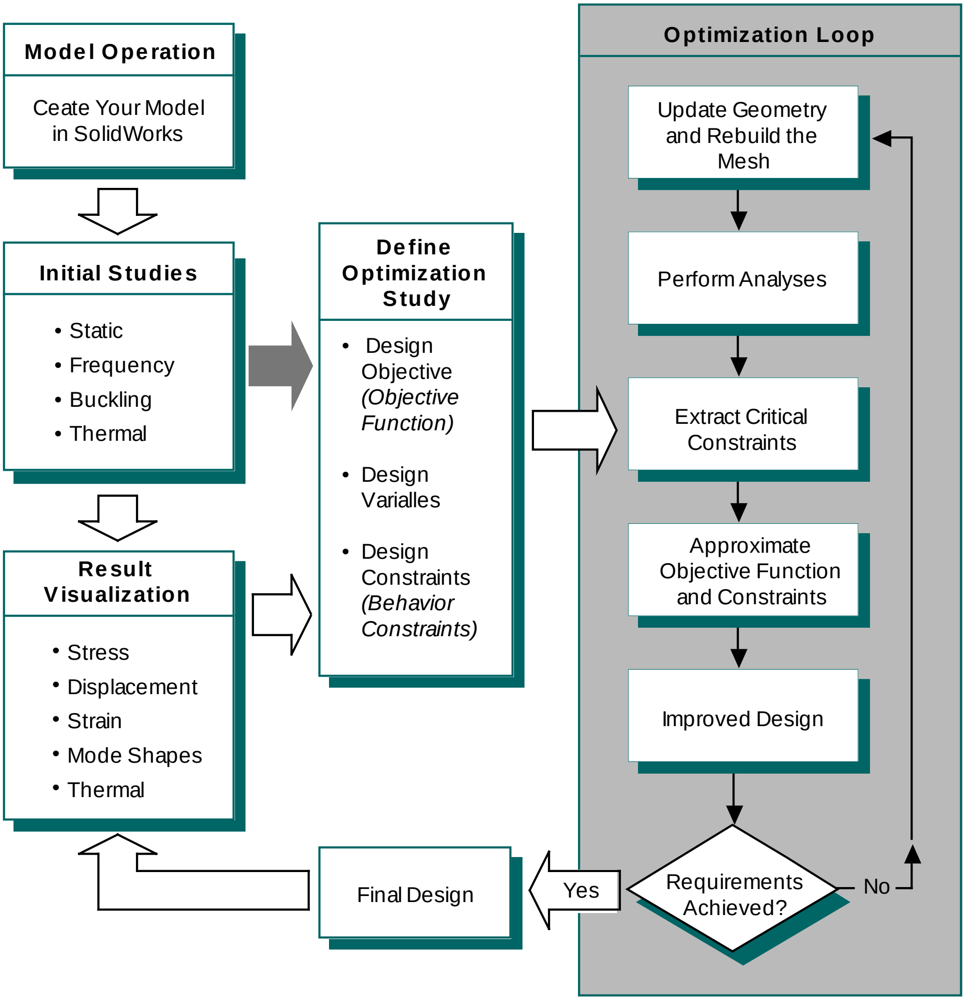
Searching for the Optimum Solution

While analysis helps you simulate a product development cycle on the computer quickly and inexpensively, you still need to create many studies and simulate many scenarios. Each time you make a change, you need to run the analysis and examine the results.

Even in a relatively simple design, there can be several dimensions that can be changed. Deciding on what combinations to try and the associated bookkeeping and result viewing can become cumbersome.

An optimization study exploits the parametric, feature-based modeling, and the automatic regeneration capabilities of SolidWorks to automate the optimization process. COSMOSWorks is equipped with a technology that quickly detects trends and identify the optimum solution in the least number of runs.

The chart below depicts the optimization process.



Using Optimization Studies

The optimization of a design requires the following sequence of steps:

- Defining and running the initial studies.
- Evaluating the results of the initial studies.
- Defining the optimization study.
- Running the optimization study.
- Viewing the results of the optimization study.
- Checking final results

Defining and Running the Initial Studies

An optimization study requires the creation of at least one initial study. Initial studies represent the basis of the optimization study. During each optimization loop, the program runs these studies with modified dimensions. The required initial studies depend on the objective function and the constraints you select.

For example, while an objective function to minimize the volume or weight does not require an initial study of a particular type, an objective function to minimize the frequency requires an initial frequency study.

The same rule applies to constraints. Each constraint you specify must be associated with a compatible initial study. For example, to define constraints on stress, frequency, and temperature, you must define static, frequency, and thermal studies.

Design scenarios are ignored by optimization studies.

After creating your model and dimensioning it to the best of your knowledge, create the initial studies and define their properties, materials, loads, and restraints. It is not recommended to use more than one study of the same type in an optimization problem.

Evaluating the Results of Initial Studies

Evaluating the results of the initial studies helps you define the optimization study properly. In particular, it is useful to examine the quantities that you want to use as constraints.

Defining the Optimization Study

To create an optimization study, right-click the document icon (top icon) in the COSMOSWorks Manager tree and select **Study**. Give a name for the study and select the **Optimization** analysis type. Note that you do NOT need to specify the mesh type as it is defined by the initial studies. You can specify the maximum allowable number of optimization loops in the properties of the optimization study.

An optimization study is defined by an objective function, design variables, and constraints. For example, you can vary dimensions in your model to seek the least possible material while stresses cannot exceed a stress limit. In this case, reducing volume is the objective function, the varying dimensions are the design variables, and the stress limit is the behavior constraint.

Materials, restraints, and loads should be consistent for all initial studies used in an optimization study.

For each optimization study, the program creates three icons named **Objective**, **Design Variables**, and **Constraints**, and a **Report** folder in the COSMOSWorks Manager tree.

Objective

The objective (or objective function) defines the goal of the optimization process. You can only specify one objective in an optimization study. The objective can be one of the following:

- minimizing volume or mass

For parts, minimizing the volume is identical to minimizing the weight since a part is made of one material. For assemblies, minimizing volume and weight can be different if parts are made of materials with different densities.

- maximizing buckling load factor
- maximizing or minimizing the resonant frequency

To define the objective function, right-click the **Objective** icon, select **Edit/Define** and then click **Add**.

Design Variables

Design variables are the changeable dimensions of the model. Any dimension can be defined as a design variable. For each design variable, you need to specify the lower and upper bounds as well as a tolerance.

When specifying the bounds of design variables, make sure that the model can regenerate without problem. The bounds of the design variables must be compatible with relations specified in SolidWorks.

You can define up to **25** design variables in an optimization study.

To define design variables, right-click the **Design Variables** icon, select **Edit/Define**, choose a dimension, and then click **Add**. If you do not see dimensions of the model to select, double-click the model in the graphics area.

Constraints

Constraints, also called behavior constraints, define the conditions that the optimized design must satisfy.

A constraint is associated with an initial study and can be defined as a component of one of the following types:

For static studies

- Nodal or element stresses
- Strains
- Displacements

For buckling studies

- First through tenth buckling load factors

For frequency studies

- First through tenth resonant Frequencies

For thermal studies

- Temperatures
- Temperature gradients
- Heat fluxes

To define constraints, right-click the **Constraints** icon and select **Edit/Define** then click **Add**. A constraint is defined by a lower bound, an upper bound, and a tolerance. You can define up to **60** constraints in an optimization study.

Running the Optimization Study

To run an optimization study, right-click its icon in the COSMOSWorks Manager tree and select **Run**. COSMOSWorks starts the optimization loops. Each loop is based on a set of values for the design variables. In each loop, COSMOSWorks runs all studies associated with the objective function and the constraints. The optimization loops terminate when the optimum design is found or the maximum allowable number of loops is reached.

Viewing Results of Optimization Study

After running the optimization study, COSMOSWorks automatically creates result icons in the COSMOSWorks Manager tree. These icons are: **Design Cycle Result**, **Design History Graph**, and **Design Local Trend Graph**.

Design Cycle Result

This folder includes icons for the final and initial designs. To show the final model, double-click the **Final Design** icon. To create an icon for a specific iteration (loop), right-click the **Design Cycle Result** folder and select **Define**.

To display detailed information, right-click the desired icon and select **Details**.

Design History Graph

You can graph the objective function, design variables, or design constraints versus optimization loops. To display the default graph, double-click the icon in the folder. To create a new graph, right-click the folder and select **Define**.

Design Local Trend Graph

You can graph a design variable versus the objective function or a selected constraint. To display the default graph, double-click the icon in the folder. To create a new graph, right-click the folder and select **Define**.

Checking the Final Results

The dimensions resulting from the optimization study may not be practical for manufacturing. You may need to round off the dimensions and check the final model.

1. Round off the design variables to comply with manufacturing standards.
2. Regenerate the model.
3. Mesh the model.
4. Run the initial studies
5. View the results to verify that all constraints are observed.

8

Viewing Results

You view the results after running a study. In viewing the results, you can generate plots, lists, graphs, and reports depending on the study and result types. In this chapter, you learn about the following topics.

- **Plotting Results**. Describes how to generate a result plot and result tools.
- **Listing Results**. Overview of the results that can be listed.
- **Graphing Results**. Shows you how to graph results.
- **Results of Structural Studies**. Lists results available from structural studies.
- **Results of Thermal Studies**. Lists results available from thermal studies.
- **Reports**. Explains the study report utility.
- **Stress Check**. Lists the basics of checking stress results and different criteria used in the checking.

Plotting Results

COSMOSWorks generates result folders in the COSMOSWorks Manager tree automatically after running a study successfully. The names of the result folders depend on the study type. A default plot is automatically generated in each folder and can be displayed by double-clicking the icon in the COSMOSWorks Manager tree.

You can define plots by right-clicking a result folder in the COSMOSWorks Manager tree and selecting **Define**.

Defining Plots

Plot dialog boxes have three tabs that define the plot as follows.

Property Tab

You can associate the plot with a SolidWorks named view.

Display Tab

You define the quantity to be plotted, units, reference geometry, result type (based on nodes or elements), and the plot type (Fringe, Vector, Section, and Iso). For transient thermal studies, you set the solution step number. For shells, you define the shell face (top, bottom, membrane, or bending).

Reference Geometry

The X, Y, and Z directions used in the **Display** tab refer to the selected reference geometry. This function enables you to view directional quantities in terms of any plane, axis, or coordinate system. For example, you can view stresses in the radial, tangential, and axial directions based on a reference axis. For orthotropic materials, you can view stresses in the orthotropic directions.

Fringe Plots

Fringe plots display the quantity on the boundaries.

Vector Plots

Vector plots are helpful in viewing directional components. They are particularly helpful when the direction of the plotted quantity vary from one location another as in the case of principal stresses and resultant vector quantities like resultant displacement and heat flux. You can dynamically control the appearance of a vector plot using the Vector Plot Options tool accessible by right-clicking a vector plot and selecting **Vector Plot Options**.

Section Plots

Section plots are helpful in viewing the quantity on sections inside the body. Sections are defined by cutting tools that can be planar, cylindrical, or spherical. After generating a section plot, you can use the **Clipping** tool to dynamically control the position and the orientation of the section.

Iso Plots

Iso plots are ideal for pinning down locations of extreme values and showing locations with a similar value of the plotted quantity. After generating an iso plot, you can use the **Clipping** tool to dynamically control the value in the plot.

Settings Tab

You customize the plot appearance in terms of model boundary, deformed shape, plot annotations, title, legend, and plot details.

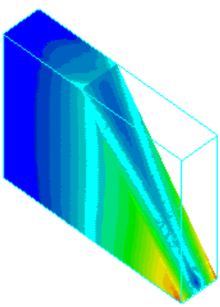
Color Map

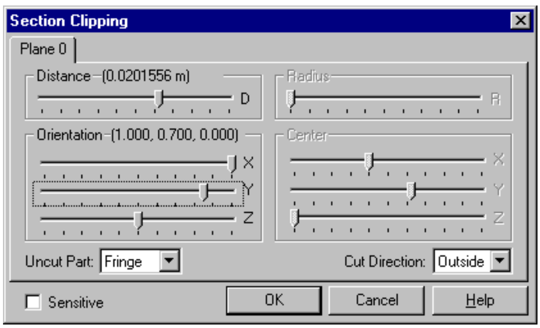
The **Color Map** dialog box lets you control the color scheme and the number of colors to be used in a plot. To access the **Color Map** dialog box, right-click a plot icon in the COSMOSWorks Manager tree and select **Color Map**. You can select a predefined color scheme or you can define your own scheme.

Clipping

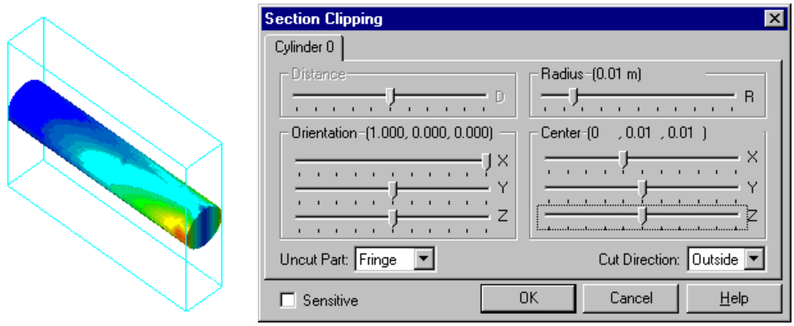
The clipping tool controls section and iso plots.

- For planar sections, the sliders control the position and the orientation of the cutting plane.

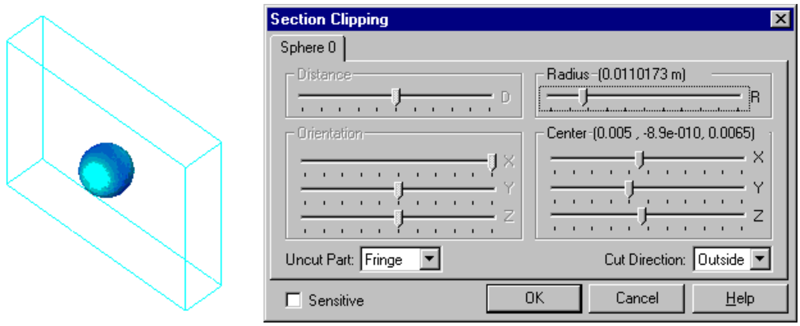




- For cylindrical sections, the sliders control the radius, orientation, and coordinates of the center of the cutting cylinder.



- For spherical sections, the sliders control the radius and the coordinates of the center of the cutting sphere.



Probing

The probing function enables you to display the numerical value of the plotted quantity at a location by clicking. Probing is available for fringe, section, and vector plots generated on undeformed shapes.

For fringe plots, the program lists the value of the plotted quantity at the closest node to the point of clicking. For section plots, you can only probe values on the sections. The program interpolates the result at the point of clicking on the section.

To start this function, click the **Probe** tool on the COSMOSWorks Result Tools toolbar, or right-click a plot icon in the COSMOSWorks Manager tree and select **Probe**. The **Probe** dialog box allows you to graph the data and save it to a file. For transient studies, you can generate time history response graphs. To highlight a location in the graphics area, select it in the dialog box.

Probing is not available for element quantities, like element stresses and strains, in this release.

Listing Results

Although plotting is usually a better result viewing tool, it is sometimes preferable to list the results in a tabular form. For example, after running a frequency study, you can list all requested resonant frequencies. Listing is available by clicking **COSMOSWorks, List Results** and selecting a menu item. You can select the following items:

Displacement

To list displacements for static, frequency, and buckling studies. For frequency and buckling studies, displacements correspond to mode shapes.

Stress

To list stresses for static studies.

Strain

To list strains for static studies.

Mode Shape

To list resonant frequencies for frequency studies or critical load factors for buckling studies.

Thermal

To list results of thermal studies.

You can list results of your analysis in the desired coordinate system by selecting the appropriate reference geometry. If you do not select a reference geometry, the program lists the results in the global coordinate system.

You can also list some other analysis related quantities by clicking **COSMOSWorks**, **Result Tools** and selecting a menu item. You can select the following items:

List Selected

The **List Selected** tool lists the plotted quantity on selected geometry (vertices, edges, and faces). The sum, average, maximum, and minimum values are also computed. To start this function, click the **List Selected** tool on the COSMOSWorks Result Tools toolbar, or right-click a plot icon in the COSMOSWorks Manager tree and select **List Selected**. To highlight a location in the graphics area, select it in the dialog box. To save the listed results to an Excel spreadsheet or to a text file, click **Save**.

Using this tool on heat flux plots for thermal problems, lists the total heat flow across selected faces.

Reaction Forces

The **Reaction Force** dialog box lets you list reaction forces on selected entities for static studies. You can access the **Reaction Force** dialog box by right-clicking the **Displacement** folder and selecting **Reaction Force**.

Contact/Friction Forces

Contact and friction forces develop on faces of components with a contact condition. The **Contact/Friction Force** dialog box lets you list contact and friction forces on selected entities for static studies. You can access the **Contact/Friction Force** dialog box by right-clicking the **Stress** folder and selecting **Contact/Friction Force**.

Graphing Results

Graphs of Probed Results

After using the **Probe** tool to identify the plotted result at the desired locations, you can graph the results. The graph assumes linear variation and equal distances between selected locations. You can use this functionality with any plot. For transient studies, you can generate time history response graphs by clicking **Response**. The response graph is generated for the full solution time regardless of the time step used to generate the plot.

To use the **Probe** tool, right-click a plot icon in the COSMOSWorks Manager tree and select **Probe**.

Graph Results on a Selected Edge

You can generate a 2D graph of the desired result on a selected edge of your model. The graph displays the result (Y-axis) versus a parametric distance along the selected edge (X-axis). This option is only available when you select one edge only. To graph the desired result on a model edge, list the desired result on the selected edge using the **List Selected** tool, then click **Plot** in the **List Selected** dialog box. You can reverse the parametric variable on the edge, click **Flip edge plot**.

Graphs for Adaptive Methods

After running a static study using an adaptive method, you can generate convergence graphs by right-clicking the study icon and selecting **Convergence Graph**.

Graphs for Design Scenarios

You can graph design scenario results after running design scenarios for a static, frequency, buckling, or thermal study. To graph design scenario results, right-click the **Design Scenario Results** folder and select **Define Graph**.

Graphs for Optimization Studies

After running an optimization studies, you can graph results obtained in optimization loops. For example, you can generate **Design History Graphs** and **Design Local Trend Graphs**. To generate an optimization graph, right-click the respective result folder and select **Define**.

Results of Structural Studies

Stress

This folder contains a nodal von Mises plot by default. To plot the default von Mises stress, double-click the icon in the **Stress** folder in the COSMOSWorks Manager tree. Most stress quantities are available based on nodes and elements. Stress error is based on elements only.

Stress Quantities

You can define the following stress quantities based selected reference geometry:

SX	= Normal stress in the X-direction
SY	= Normal stress in the Y-direction
SZ	= Normal stress in the Z-direction
TXY	= Shear stress in the Y-direction in the plane normal to X
TXZ	= Shear stress in the Z-direction in the plane normal to X
TYZ	= Shear stress in the Z-direction in the plane normal to Y

The following quantities do not use reference geometry:

P1	= Normal stress in the first principal direction
P2	= Normal stress in the second principal direction
P3	= Normal stress in the third principal direction
VON	= von Mises stress
INT	= Stress intensity = P1 - P3

ERR = Relative error in stresses (available for *Element stresses* only)

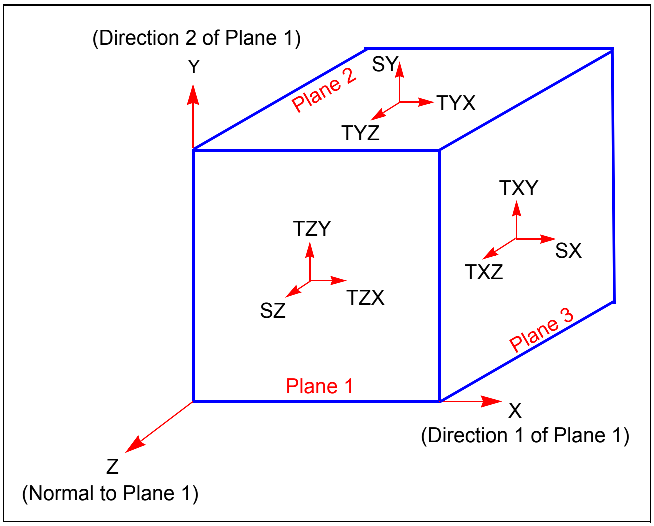
Where:

$$VON = \{(1/2)[(SX-SY)^2+(SX-SZ)^2+(SY-SZ)^2]+3(TXY^2+TXZ^2+TYZ^2)\}^{(1/2)}$$

VON is expressed in terms of the principal stresses P1, P2, and P3 as given below:

$$VON = \{(1/2)[(P1-P2)^2+(P1-P3)^2+(P2-P3)^2]\}^{(1/2)}$$

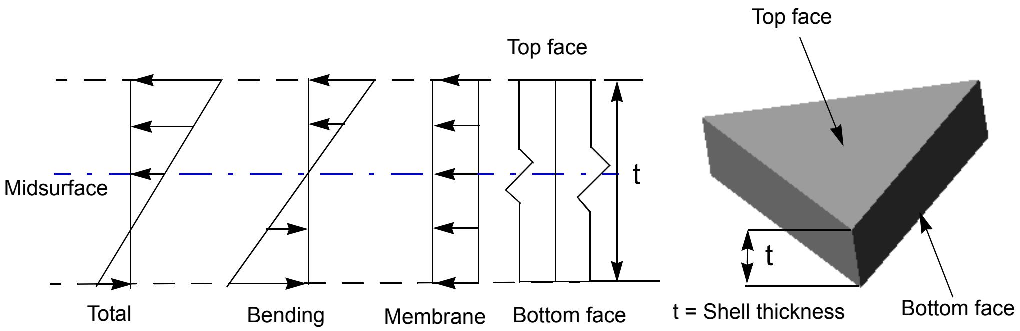
The notation of stress components is shown in the figure.



Stresses for Shell Models

A shell element has a top face and a bottom face. Stresses on the top faces are different from the stresses on the bottom face unless the shell is under pure membrane forces. In pure bending, the stresses on the top and bottom faces are equal in magnitude but opposite in the direction (one face in compression and the other is tension).

Stress results of shell elements can be decomposed into membrane and bending components as shown in the figure:



Membrane Stresses

Membrane stresses are induced by loads acting in the plane of the shell. They are tensile or compressive uniform stresses across the thickness of an element.

Bending Stresses

Bending stresses are induced by pure bending loads. They vary linearly across the thickness of a shell element. They have the same magnitude and opposite directions (tensile or compressive) at the top and bottom faces.

The following options are available:

Top: total (membrane + bending) stresses at the top face

Bottom: total (membrane + bending) stresses at the bottom face

Membrane: membrane stress component

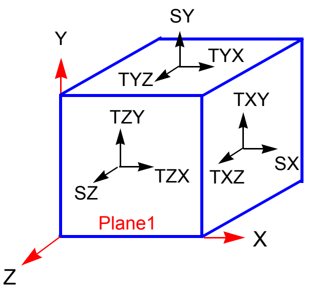
Bending: bending stress component

Faces of adjacent shells have to be aligned before running the analysis. We strongly recommend that you view the stresses on both faces for proper assessment of stress results. Refer to the *Meshing* chapter for more information on shell elements.

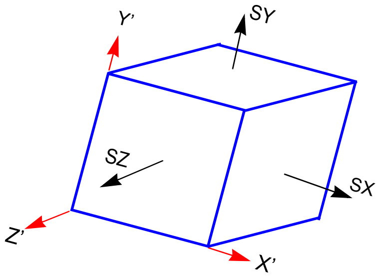
Plotting Principal Stresses

The state of stresses at a point is completely defined by normal and shear stress components in reference to plane, axis, or a coordinate system XYZ. In general, the values of the stress components change if the coordinate system is rotated. At a certain orientation (X’Y’Z’), all shear stresses vanish and the state of stresses is completely defined by three normal stress components. These three normal stress components are referred to as principal stresses and the corresponding reference axes (X’Y’Z’) are referred to as principal axes.

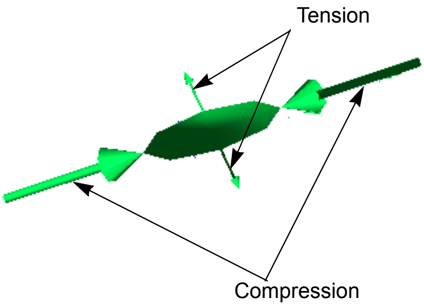
You can plot the three principal stresses at once. The principal stresses at a node or element center are represented by an ellipsoid. The radii of the ellipsoid represent the magnitudes of the three principal stresses. The direction of the stress (tension/ compression) is represented by arrows. The color code is based on the von Mises stress values, a scalar quantity. If one of the principal stresses is zero, the ellipsoid becomes a planar ellipse. If the three principal stresses have the same magnitude, the ellipsoid becomes a sphere. In the case of simple uniaxial stress, the ellipsoid becomes a line.



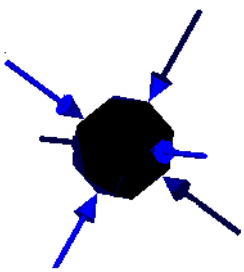
Stress components



Shear stresses vanish



One of the principal stresses is zero or very small in this case



The three principal stresses are in compression

To plot principal stresses, define a vector von Mises plot.

Displacement

Displacements refer to the movement of nodes. This folder includes a fringe plot of the resultant displacement by default.

Displacement Quantities

You can define the following plots in terms of a reference geometry:

UX	= Displacement in the X-direction
UY	= Displacement in the Y-direction
UZ	= Displacement in the Z-direction
URES	= Resultant displacement (does not use the reference geometry)
RFX	= Reaction force in the X-direction
RFY	= Reaction force in the Y-direction
RFZ	= Reaction force in the Z-axis
RFRES	= Resultant reaction force (does not use the reference geometry)

where:

$$\mathrm{URES} = \sqrt{\mathrm{UX}^2 + \mathrm{UY}^2 + \mathrm{UZ}^2}$$

and

$$\mathrm{RFRES} = \sqrt{\mathrm{RFX}^2 + \mathrm{RFY}^2 + \mathrm{RFZ}^2}$$

Deformed Shape

A deformed shape plot shows the predicted shape of the model in the specified environment. Stress, strain, and displacement results can be plotted on the deformed or deformed shapes. The displacement plot is more informative than the deformation plot because it contains the color code.

Deformation scale

In most cases, the deformation of the model is very small compared to the dimensions of the model. If plotted to scale, the deformed and undeformed shape approximately coincide. When plotting a quantity on the deformed shape, COSMOSWorks scales the maximum displacement to 10% of the diagonal of the bounding box of the model. While this is not real, it is useful in illustrating the trend of motion.

In viewing the results of static studies with contact conditions, this artificial amplification can lead to confusion. In such cases, it is important to use a scale factor of 1.0 to depict the real deformed shape.

Strain

This folder contains a plot of equivalent element strains by default. To plot the default equivalent strain, double-click the icon in the Strain folder in the COSMOSWorks Manager tree. Strain quantities are available for elements only. Nodal strains are not available. Each element is shown in a uniform color.

Strain Quantities

You can define the following strain quantities based on the reference geometry plots:

EPSX	= Normal strain in the X-direction)
EPSY	= Normal strain in the Y-direction
EPSZ	= Normal strain in the Z-direction
GMXY	= Shear strain in the Y-direction in the plane normal to X
GMXZ	= Shear strain in the Z-direction in the plane normal to X
GMYZ	= Shear strain in the Z-direction in the plane normal to Y
ESTRN	= Equivalent strain
SEDENS	= Strain energy density
ENERGY	= Total strain energy
E1	= Normal strain in the first principal direction
E2	= Normal strain in the second principal direction
E3	= Normal strain in the third principal direction

Where:

$$\mathrm{ESTRN} = 2.0\sqrt{\frac{\varepsilon_1 + \varepsilon_2}{3.0}}$$

$$\varepsilon_1 = 0.5[(\mathrm{EPSX} - \mathrm{meanstrain})^2 + (\mathrm{EPSY} - \mathrm{meanstrain})^2 + (\mathrm{EPSZ} - \mathrm{meanstrain})^2]$$

$$\varepsilon_2 = 0.25[\mathrm{GMXY}^2 + \mathrm{GMXZ}^2 + \mathrm{GMYZ}^2]$$

$$\mathrm{meanstrain} = (\mathrm{EPSX} + \mathrm{EPSY} + \mathrm{EPSZ})/3$$

Results of Thermal Studies

The **Thermal** folder in a thermal study contains a nodal temperature plot by default. To plot the default nodal temperature, double-click the icon in the **Thermal** folder in the COSMOSWorks Manager tree.

You can plot results from steady state or transient thermal analysis.

Thermal Quantities

Sets the thermal component to be plotted. Directions of vector quantities are based on the selected reference geometry. The following components are available:

TEMP	= Nodal temperature
GRADX	= Temperature gradient in the X-direction
GRADY	= Temperature gradient in the Y-direction
GRADZ	= Temperature gradient in the Z-direction
GRADN	= Resultant temperature gradient
HFLUXX	= Heat flux in the X-direction
HFLUXY	= Heat flux in the Y-direction
HFLUXZ	= Heat flux in the Z-direction
HFLUXN	= Resultant heat flux

Where:

$$\mathrm{GRADN} = \sqrt{\mathrm{GRADX}^2 + \mathrm{GRADY}^2 + \mathrm{GRADZ}^2}$$

$$\mathrm{HFLUXN} = \sqrt{\mathrm{HFLUXX}^2 + \mathrm{HFLUXY}^2 + \mathrm{HFLUXZ}^2}$$

Reports

The **Report** tool helps you document your studies quickly and systematically by generating internet-ready reports. The reports are structured to describe all aspects of the study.

Plots created in the COSMOSWorks Manager tree can be included automatically in the report. You can also insert images, animations (AVI videos), and VRML files in the report. A printer-friendly version of the report can be generated automatically. Reports provide an excellent way to share study results with others online or in printed format. You can modify the various sections of the report by inserting text or graphics.

To share a report, send all associated image files along with the html files. The receiver should place all files in the same folder for viewing.

For static, frequency, buckling, and thermal studies with design scenarios, the report automatically includes information, results, and graphs related to design scenarios.

To start the **Report** wizard, right-click the **Report** folder of the study and select **Define**. Settings that you enter in the **Report** wizard are used for the report only. For example if you change the **Result file location** in the **Set File** section, the actual result location does not change.

Stress Check

After running a static study, you can use the **Design Check Wizard** to assess the safety of your design. COSMOSWorks provides several criteria to calculate the factor of safety distribution.

Factor of Safety

The **Design Check Wizard** evaluates the factor of safety at each node based on a failure criterion. You can plot the factor of safety distribution throughout the model, or you can just plot regions of the model with a factor of safety smaller than a specified value to identify weak areas of the design. Large factors of safety in a region indicate that you can save material from that region. Many codes require a minimum factor of safety between 1.5 and 3.0.

Factors of Safety Interpretation

- A factor of safety less than 1.0 at a location indicates that the material at that location has failed.
- A factor of safety of 1.0 at a location indicates that the material at that location has just started to fail.
- A factor of safety larger than 1.0 at a location indicates that the material at that location is safe.
- The material at a location will start to fail if you apply new loads equal to the current loads multiplied by the resulting factor of safety.

Non-Dimensional Stress Distribution (Stress Ratio)

You can use the **Design Check Wizard** to plot a quantity called the non-dimensional stress distribution defined as the inverse of the factor of safety.

$$\textit{Non-dimensional stress} = \frac{1}{\mathrm{FOS}}$$

Failure Criteria

Failure criteria predict the failure of a material subjected to a state of stresses. The Design Check Wizard assesses the safety of the model based on a selected failure criterion.

A material may behave in a ductile or brittle manner depending on the temperature, rate of loading, chemical environment, and the formation process. No single failure criterion is best applicable to a material under all conditions. You should use all information available to you about the material to select a failure criterion. You can try more than one criterion to assess the safety of the model.

COSMOSWorks provides the following failure criteria:

- **Maximum von Mises Stress Criterion**,
- **Maximum Shear Stress Criterion**,
- **Mohr-Coulomb Stress Criterion**, and
- **Maximum Normal Stress Criterion**

Maximum von Mises Stress Criterion

The maximum von Mises stress criterion is based on the von Mises-Hencky theory, also known as the Shear-energy theory or the Maximum distortion energy theory.

In terms of the principal stresses σ_1, σ_2, and σ_3, the von Mises stress is expressed as:

$$\sigma_{vonMises} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}}$$

The theory states that a ductile material starts to yield at a location when the von Mises stress becomes equal to the stress limit. In most cases, the yield strength is used as the stress limit. However, COSMOSWorks allows you to use the ultimate tensile or set your own stress limit.

$$\sigma_{vonMises} \geq \sigma_{limit}$$

Yield strength is a temperature-dependent property. This specified value of the yield strength should consider the temperature of the component. The factor of safety at a location is calculated from:

$$\textit{Factor of Safety (FOS)} = \frac{\sigma_{limit}}{\sigma_{vonMises}}$$

Pure Shear

In the case of pure shear τ, von Mises stress can be expressed as:

$$\sigma_{vonMises} = \sqrt{3}\tau$$

Failure occurs if:

$$\tau_{max} = 0.577\sigma_{yield}$$

Maximum Shear Stress Criterion

The maximum shear stress criterion, also known as Tresca yield criterion, is based on the Maximum Shear stress theory.

This theory predicts failure of a material to occur when the absolute maximum shear stress (τ_{max}) reaches the stress that causes the material to yield in a simple tension test. The Maximum shear stress criterion is used for ductile materials.

$$\tau_{max} \geq \frac{\sigma_{limit}}{2}$$

τ_{max} is the greatest of τ_{12} , τ_{23} and τ_{13}

Where:

$$\tau_{12} = \frac{\sigma_1 - \sigma_2}{2} \;;\quad \tau_{23} = \frac{\sigma_2 - \sigma_3}{2};\quad \tau_{13} = \frac{\sigma_1 - \sigma_3}{2}$$

Hence:

Factor of safety (FOS) = $\dfrac{\sigma_{limit}}{2\tau_{max}}$

Comparing the von Mises and Tresca Stress Criteria

The maximum shear stress criterion is more conservative than the von Mises stress criterion since the hexagon representing the shear stress criterion is enclosed within the ellipse representing the von Mises stress criterion.

For a condition of pure shear, von Mises stress criterion predicts failure at (0.577*yield strength) whereas the shear stress criterion predicts failure at 0.5 yield strength. Actual torsion tests used to develop pure shear have shown that the von Mises stress criterion gives more accurate results than the maximum shear stress theory.

Mohr-Coulomb Stress Criterion

The Mohr-Coulomb stress criterion is based on the Mohr-Coulomb theory also known as the Internal Friction theory. This criterion is used for brittle materials with different tensile and compressive properties. Brittle materials do not have a specific yield point and hence it is not recommended to use the yield strength to define the limit stress for this criterion.

This theory predicts failure to occur when:

$$\sigma_1 \geq \sigma_{TensileLimit} \quad \textit{if } \sigma_1 > 0 \quad \& \quad \sigma_3 > 0$$

$$\sigma_3 \geq -\sigma_{CompressiveLimit} \quad \textit{if } \sigma_1 < 0 \quad \& \quad \sigma_3 < 0$$

$$\frac{\sigma_1}{\sigma_{TensileLimit}} + \frac{\sigma_3}{\sigma_{CompressiveLimit}} < 1 \quad \textit{if } \sigma_1 \geq 0\ , \quad \sigma_3 \leq 0$$

The factor of safety is given by:

$$\textit{Factor of safety} = \left\{ \frac{\sigma_1}{\sigma_{TensileLimit}} + \frac{\sigma_3}{\sigma_{CompressiveLimit}} \right\}^{-1}$$

Maximum Normal Stress Criterion

The maximum normal stress criterion also known as Coulomb's criterion is based on the Maximum normal stress theory. According to this theory failure occurs when the maximum principal stress reaches the ultimate strength of the material for simple tension.

This criterion is used for brittle materials. It assumes that the ultimate strength of the material in tension and compression is the same. This assumption is not valid in all cases. For example, cracks decrease the strength of the material in tension considerably while their effect is far less smaller in compression because the cracks tend to close.

Brittle materials do not have a specific yield point and hence it is not recommended to use the yield strength to define the limit stress for this criterion.

This theory predicts failure to occur when:

$$\sigma_1 \geq \sigma_{limit}$$

where: σ_1 is the maximum principal stress.

Hence:

$$\textit{Factor of safety} = \frac{\sigma_{limit}}{\sigma_1}$$

Summary of Failure Criteria

Criterion	Factor of Safety	Material Type
von Mises stress	$\frac{\sigma_{limit}}{\sigma_{vonMises}}$	Ductile
Shear stress (Tresca)	$\frac{\sigma_{limit}}{2\tau_{max}}$	Ductile
Mohr-Coulomb stress	$\frac{\sigma_{TensileLimit}}{\sigma_1}$ if $\sigma_1 > 0$ & $\sigma_3 > 0$ $-\frac{\sigma_{CompressiveLimit}}{\sigma_3}$ if $\sigma_1 < 0$ & $\sigma_3 < 0$ $\frac{\sigma_{TensileLimit}\sigma_{CompressiveLimit}}{\sigma_1\sigma_{CompressiveLimit} + \sigma_3\sigma_{TensileLimit}}$ if $\sigma_1 \geq 0$ & $\sigma_3 \leq 0$	Brittle materials with different tensile and compressive strengths
Maximum Normal Stress (Principal)	$\frac{\sigma_{limit}}{\sigma_1}$	Brittle

Using Design Check for Assemblies

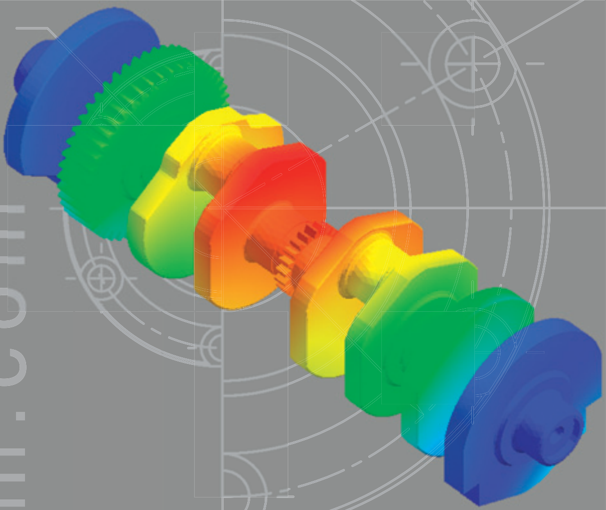
When using the **Design Check Wizard** to check the safety of assemblies, you can choose to check a single component or all components at once. If you choose to check a single component, the program uses the specified stress limit for its material.

If you choose to check all components in the assembly at once, the program uses the yield strength of the material of first component that appears in the COSMOSWorks Manager tree for all components. In all cases, you can specify your own stress limit.

You can check a number of components that have the same material in one step by hiding other components and performing design check on ALL components.

Index

www.cosmosm.com



www.cosmosm.com

COSMOS Analysis Products
Tel: +1-800-469-7287
Fax: +1-310-207-7805
Email: info@cosmosm.com
www.cosmosm.com

CWMISENG0703