

SOLIDWORKS SIMULATION 2015

Black Book

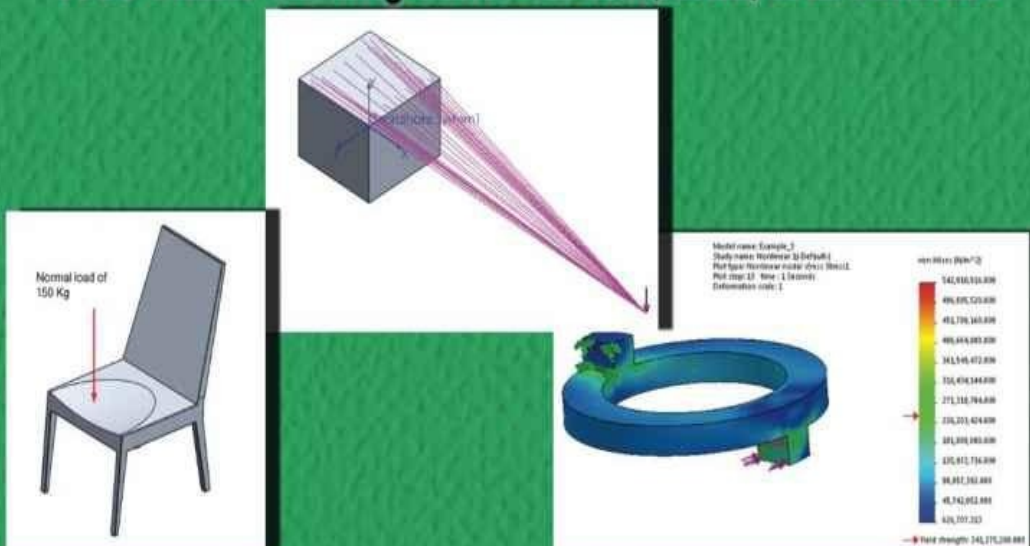
Revised & Updated

Book covers:

FEA Fundamentals and theory, Static Analysis, Non-Linear Static Analysis, Non-Linear Dynamic Analysis, Linear Dynamic Analysis, Thermal Analysis, Frequency Analysis, Buckling Analysis, Fatigue Analysis, Drop Test, Pressure Vessel Design, Design Study

Resources available with the book:

- * Free part files and tutorial files for students
- * Exercise files and video tutorials on desired topics
- * Project for download
- * Free online training demo on CAD/CAM/CAE software



Matt Weber
Gaurav Verma

SOLIDWORKS SIMULATION 2015

BLACK BOOK

Matt Weber
CAD/CAM/CAE Expert

Gaurav Verma
Technical Editor



Published by CADCAMCAEWORKS, USA. Copyright © 2015. All rights reserved. No part of this publication may be reproduced or distributed in any form or by any means, or stored in the database or retrieval system without the prior permission of CADCAMCAEWORKS. To get the permissions, contact at cadcamcaeworks@gmail.com

ISBN # 978-1-505-90058-3

NOTICE TO THE READER

Publisher does not warrant or guarantee any of the products described in the text or perform any independent analysis in connection with any of the product information contained in the text. Publisher does not assume, and expressly disclaims, any obligation to obtain and include information other than that provided to it by the manufacturer.

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly assumes all risks in connection with such instructions.

The Publisher makes no representation or warranties of any kind, including but not limited to, the warranties of fitness for a particular purpose or merchantability, nor are any such representations implied with respect to the material set forth herein, and the publisher takes no responsibility with respect to such material. The publisher shall not be liable for any special, consequential, or exemplary damages resulting, in whole or part, from the reader's use of, or reliance upon, this material.

DEDICATION

To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations

To students, who are the future of the world

THANKS

To my friends and colleagues

To my family for their love and support

Training and Consultant Services

At CAD/CAM/CAE WORKS, we provide effective and affordable one to one online training on various software packages in Computer Aided Design(CAD), Computer Aided Manufacturing(CAM), Computer Aided Engineering (CAE), Computer programming languages(C/C++, Java, .NET, Android, Javascript, HTML and so on). The training is delivered through remote access to your system and voice chat via Internet at any time, any place, and at required pace to individuals, groups, students of colleges/universities, and CAD/CAM/CAE training centers. The main features of this program are:

Training as per your need

Highly experienced Engineers and Technicians conduct the classes on the software applications used in the industries. The methodology adopted to teach the software is totally practical based, so that the learner can adapt to the design and development industries in almost no time. The efforts are to make the training process cost effective and

time saving while you have the comfort of your time and place, thereby relieving you from the hassles of traveling to training centers or rearranging your time table.

Current Software Packages on which we provide basic and advanced training are:

CAD/CAM/CAE: CATIA, Creo Parametric, Creo Direct, SolidWorks, Autodesk Inventor, Solid Edge, UG NX, AutoCAD, AutoCAD LT, EdgeCAM, MasterCAM, SolidCAM, DelCAM, BOBCAM, UG NX Manufacturing, UG Mold Wizard, UG Progressive Die, UG Die Design, SolidWorks Mold, Creo Manufacturing, Creo Expert Machinist, NX Nastran, Hypermesh, SolidWorks Simulation, Autodesk Simulation Mechanical, Creo Simulate, Gambit, ANSYS and many others.

Computer Programming Languages: C++, VB.NET, HTML, Android, Javascript and so on.

Game Designing: Unity.

Civil Engineering: AutoCAD MEP, Revit Structure, Revit Architecture, AutoCAD Map 3D and so on.

We also provide consultant services for Design and development on the above mentioned software packages

For more information you can mail us at:

cadcamcaeworks@gmail.com

Table of Contents

[SolidWorks Simulation 2015](#)

[Black Book](#)

[Training and Consultant Services](#)

[Preface](#)

[About Author](#)

[Introduction to](#)

[Simulation](#)

[Chapter 1](#)

[Simulation](#)

[Types of Analyses performed in SolidWorks Simulation](#)

[Static Analysis](#)

[Dynamic Analysis](#)

[Thermal analysis](#)

[Steady State Thermal Analysis](#)

[Transient Thermal Analysis](#)

[Drop Test Studies](#)

[Fatigue Analysis](#)

[Pressure Vessel Design Study](#)

[Design Study](#)

[FEA](#)

[Starting SolidWorks Simulation](#)

[On clicking Study Advisor](#)

[On clicking New Study](#)

[Page left blank intentionally](#)

[Basics of](#)

[Analyses](#)

[Chapter 2](#)

[Starting Analysis](#)

[Applying Material](#)

[Adding Custom Materials](#)

[Defining Fixtures](#)

[Fixtures Advisor](#)

[Fixed Geometry](#)

[Roller/Slider](#)

[Fixed Hinge](#)

[Elastic Support](#)

[Advanced Fixtures](#)

[External Load Advisor](#)

[External Load Advisor](#)

[Force](#)

[Torque](#)

[Pressure](#)

[Gravity](#)

[Centrifugal Force](#)

[Bearing Load](#)

[Remote Loads/Mass](#)

[Distributed Mass](#)

[Temperature](#)

[Flow Effects/Thermal Effects](#)

[Connections Advisor](#)

[Connections Advisor](#)

[Contact Sets](#)

[Component Contact](#)

[Contact Visualization Plot](#)

[Spring](#)

[Pin](#)

[Bolt](#)

[Spot Weld](#)

[Edge Weld](#)

[Link](#)

[Bearing](#)

[Rigid Connection](#)

[Meshing](#)

[Running Analysis](#)

[Results Advisor](#)

[Results Advisor](#)

[Model Only\(No Results\)](#)

[New Plot](#)

[List Stress, Displacement and Strain](#)

[Result Force](#)

[Static](#)

[Analysis](#)

[Chapter 3](#)

[Linear static analysis](#)

[Assumptions](#)

[Performing Static Analysis](#)

[Starting the Static Analysis](#)

[Applying Material](#)

[Applying Fixtures](#)

[Applying Loads](#)

[Meshing the model](#)

[Running the Analysis](#)

[Results](#)

[Optimizing product](#)

[Shell Manager](#)

[Performing Static Analysis Using Study Advisor](#)

[Starting the Static Analysis](#)

[Non-Linear](#)

[Static Analysis](#)

[Chapter 4](#)

[Non Linear Static Analysis](#)

[Starting Nonlinear Static Analysis](#)

[Applying Material](#)

[Applying Fixtures](#)

[Applying Time Varying Force](#)

[Running the analysis](#)

[Performing Nonlinear Static analysis on an assembly](#)

[Starting Non-Linear Static Analysis](#)

[Applying the Material](#)

[Applying Fixtures](#)

[Applying Forces](#)

[Applying Connections](#)

[Running the analysis](#)

[Performing Nonlinear Static analysis on a ring under application of Torque](#)

[Starting Non-Linear Static Analysis](#)

[Applying the Material](#)

[Applying Fixtures](#)

[Applying Forces](#)

[Running the analysis](#)

[Non-Linear](#)

[Dynamic Analysis](#)

[Chapter 5](#)

[Non Linear Dynamic Analysis](#)

[Starting Nonlinear Dynamic Analysis](#)

[Applying Material](#)

[Applying Fixtures](#)

[Applying Time Varying Force](#)

[Applying Connections](#)

[Global Damping](#)

[Running the analysis](#)

[Frequency Analysis](#)

[Chapter 6](#)

[Introduction](#)

[Starting Frequency Analysis](#)

[Applying Material](#)

[Applying Fixtures](#)

[Applying Forces](#)

[Setting Parameters for the analysis](#)

[Running the analysis](#)

[Linear](#)

[Dynamic Analysis](#)

[Chapter 7](#)

[Linear Dynamic Analysis](#)

[Modal Time History Analysis](#)

[Harmonic Analysis](#)

[Random Vibration Analysis](#)

[Response Spectrum Analysis](#)

[Starting Random Vibration Analysis](#)

[Applying Material](#)

[Applying Fixtures](#)

[Applying Force Vibrations](#)

[Applying Global Damping](#)

[Running the analysis](#)

[Modal Time History Analysis](#)

[Harmonic Analysis](#)

[Response Spectrum Analysis](#)

[Thermal Analysis](#)

[Chapter 8](#)

[Introduction](#)

[Important terms related to Thermal Analysis](#)

[Thermal Loads](#)

[Temperature](#)

[Convection](#)

[Heat Power](#)

[Heat Flux](#)

[Radiation](#)

[Contact Set](#)

[Practical on Steady State](#)

[Thermal Analysis](#)

[Starting the analysis](#)

[Applying Material](#)

[Applying temperature on the face](#)

[Applying heat generated by gas combustion on the head of the piston](#)

[Running the Analysis](#)

[Switching from steady state analysis to transient thermal analysis](#)

[Specifying Initial Temperature](#)

[Running the analysis](#)

[Using Probe to find out temperature](#)

[Buckling Analysis](#)

[Chapter 9](#)

[Introduction](#)

[Use of Buckling Analysis](#)

[Starting the Buckling Analysis](#)

[Applying Material](#)

[Applying Fixture](#)

[Applying the Force](#)

[Running the Analysis](#)

[Fatigue Analysis](#)

[Chapter 10](#)

[Introduction](#)

[Stages of Failure Due to Fatigue](#)

[Starting Fatigue Analysis](#)

[Initiating Fatigue Analysis](#)

[Applying or Editing S-N data](#)

[Modifying the Static Analysis earlier performed](#)

[Running the Fatigue Analysis](#)

[Changing Properties of Fatigue Analysis](#)

[Drop-Test](#)

[Chapter 11](#)

[Introduction](#)

[Starting Drop Test](#)

[Applying Material](#)

[Creating Mesh](#)

[Parameters for Drop](#)

[Running the Analysis](#)

[Pressure Vessel Design](#)

[and Design Study](#)

[Chapter 12](#)

[Introduction](#)

[Thermal Analysis 1](#)

[Static Analysis 1 & 2](#)

[Starting Pressure Vessel Design Study](#)

[Setup for Design Study](#)

[Running the Design Study](#)

[Design Study](#)

[Starting Design Study](#)

[Setting options for Design Study](#)

[Specifying parameters that are to be changed](#)

[Specifying restrictions for changes](#)

[Setting Goal for the study](#)

[Running the Study](#)

[Fundamentals of FEA](#)

[Chapter 13](#)

[Introduction](#)

[GENERAL DESCRIPTION OF THE METHOD](#)

[A BRIEF EXPLANATION OF FEA FOR A STRESS ANALYSIS PROBLEM](#)

[FINITE ELEMENT METHOD V/S CLASSICAL METHODS](#)

[FEM VS FINITE DIFFERENCE METHOD \(FDM\)](#)

[NEED FOR STUDYING FEM](#)

[Warning To Fea Package Users](#)

[Geometric Discontinuities](#)

[Discontinuity of Loads](#)

[Discontinuity of Boundary conditions](#)

[Material Discontinuity](#)

[REFINING MESH](#)

[Use Of Symmetry](#)

[HIGHER ORDER ELEMENTS V/S](#)

[REFINED MESH](#)

[Optimum process of FEA through](#)

[SolidWorks Simulation](#)

[Space for Student Notes](#)

[Project on Analysis](#)

[Chapter 14](#)

[Project](#)

[Starting Simulation](#)

[Performing the Static Analysis](#)

[Performing the Frequency Analysis](#)

[Performing Fatigue Study](#)

Preface

SolidWorks Simulation 2015 is an extension to SolidWorks package. Easy-to-use CAD-embedded analysis capabilities enable all designers and engineers to simulate and analyze design performance. You can quickly and easily employ advanced simulation techniques to optimize performance while you design, to cut down on costly prototypes, eliminate rework and delays, and save you time and development costs.

The SolidWorks Simulation 2015 Black Book, is written to help professionals as well as learners in performing various tedious jobs in Finite Element Analysis. The book follows a step by step methodology. This book is more concentrated on making you able to use tools at right places. The book covers almost all the information required by a learner to master the SolidWorks Simulation. The book starts with basics of FEA, goes through all the simulation tools and ends up with practical examples of analysis. Chapters on manual FEA ensure the firm understanding of FEA concepts through SolidWorks Simulation. This upgraded version of book contains our special sections named “Why?”. We have given reasons for selecting every option in analysis under these “Why?” sections. Some of the salient features of this book are :

In-Depth explanation of concepts

Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world.

Topics Covered

Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easily find the topic of his/her interest easily.

Instruction through illustration

The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 800 illustrations that make the learning process effective.

Tutorial point of view

The book explains the concepts through the tutorial to make the understanding of users firm and long lasting. Each chapter of the book has tutorials that are real world projects.

“Why?”



The book explains the reasons for selecting options or setting a parameters in tutorials explained in the book.

Project

Free projects and exercises are provided to students for practicing.

For Faculty

If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept.

Formatting Conventions Used in the Text

All the key terms like name of button, tool, drop-down etc. are kept bold.

Free Resources

Link to the resources used in this book are provided to the users via email. To get the

resources, mail us at cadcamcaeworks@gmail.com with your contact information. With your contact record with us, you will be provided latest updates and informations regarding various technologies. The format to write us e-mail for resources is as follows:

Subject of E-mail as ***Application for resources of _____ Black Book.***

Also, given your information like

Name:

Course pursuing/Profession:

Contact Address:

E-mail ID:

For Any query or suggestion

If you have any query or suggestion please let us know by mailing us on cadcamcaeworks@gmail.com. Your valuable constructive suggestions will be incorporated in our books and your name will be addressed in special thanks area of our books.

About Author

The author of this book, Matt Weber, has written many books on CAD/CAM/CAE books. He has written SolidWorks 2015 Black Book as a CAD companion for this book. When SolidWorks Simulation 2015 Black Book covers details of simulation, the SolidWorks 2015 Black Book covers all the tools and techniques of modeling. The author has hand on experience on almost all the CAD/CAM/CAE packages. Besides that he is a good person in his real life, helping nature for everyone. If you have any query/doubt in any CAD/CAM/CAE package, then you can directly contact the author by writing at cadcamcaeworks@gmail.com

The technical editor of the book, Gaurav Verma, has written books on CAM packages. One of his most selling books, Creo Manufacturing 2.0 for Designer and Machinists is available on Amazon for sale. He has also written MasterCAM X7 for SolidWorks Black Book and AutoCAD Electrical 2015 Black Book.

With the support of CreateSpace, we assure the best books and support to the education on latest technologies.

We recommend you to use SolidWorks 2015 Black Book as SolidWorks CAD companion with this book for best results.

INTRODUCTION TO SIMULATION

CHAPTER 1



Topics Covered

The major topics covered in this chapter are:

- ***Simulation.***
- ***Types of Analyses performed in SolidWorks Simulation.***
- ***FEA***
- ***User Interface of SolidWorks Simulation.***

SIMULATION

Simulation is the study of effects caused on an object due to real-world loading conditions. Computer Simulation is a type of simulation which uses CAD models to represent real objects and it applies various load conditions on the model to study the real-world effects. SolidWorks Simulation is one of the Computer Simulation programs available in the

market. In SolidWorks Simulation, we apply loads on a constrained model under predefined environmental conditions and check the result(visually and/or in the form of tabular data). The types of analyses that can be performed in SolidWorks are given next.

TYPES OF ANALYSES PERFORMED IN SOLIDWORKS SIMULATION

SolidWorks Simulation performs almost all the analyses that are generally performed in Industries. These analyses and their use are given next.

Static Analysis

This is the most common type of analysis we perform. In this analysis, loads are applied to a body due to which the body deforms and the effects of the loads are transmitted throughout the body. To absorb the effect of loads, the body generates internal forces and reactions at the supports to balance the applied external loads. These internal forces and reactions cause stress and strain in the body. Static analysis refers to the calculation of displacements, strains, and stresses under the effect of external loads, based on some assumptions. The assumptions are as follows.

- All loads are applied slowly and gradually until they reach their full magnitudes. After reaching their full magnitudes, load will remain constant (i.e. load will not vary against time).
- Linearity assumption: The relationship between loads and resulting responses is linear. If you double the magnitude of loads, for example, the response of the model (displacements, strains and stresses) will also double. You can make linearity assumption if:
 - a. All materials in the model comply with Hooke's Law that is stress is directly proportional to strain.
 - b. The induced displacements are small enough to ignore the change in stiffness caused by loading.
 - c. 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.

If the above assumptions are valid for your analysis, then you can perform **Linear Static Analysis**. For example, a cantilever beam fixed at one end and force applied on other end; refer to [Figure-1](#).

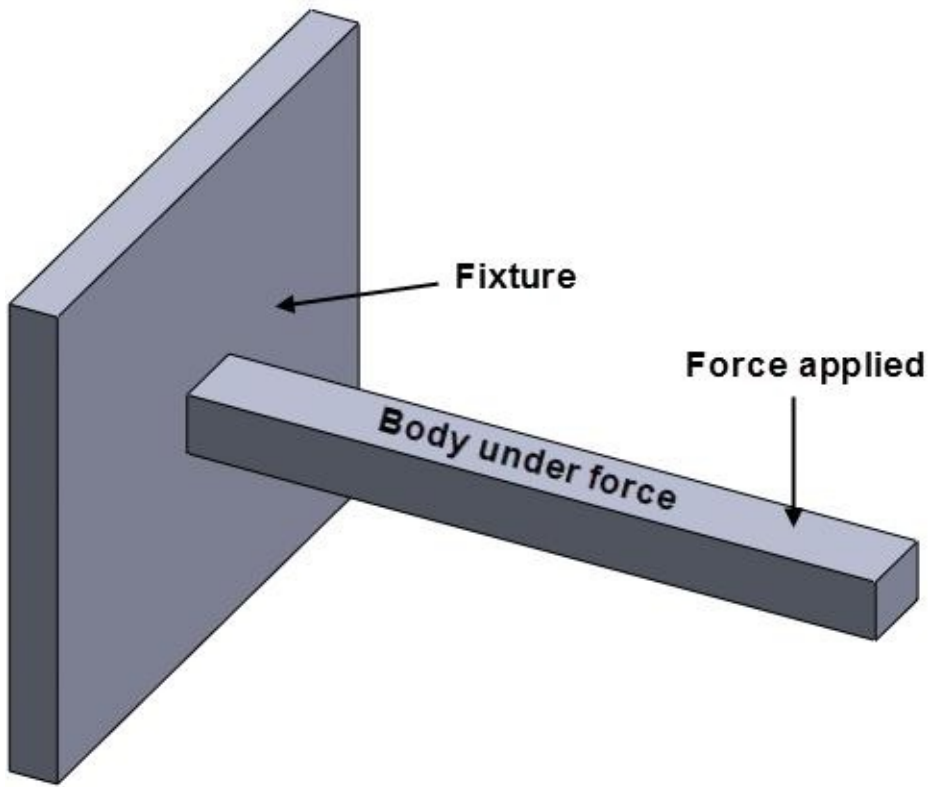


Figure-1. Linear static analysis example

If the above assumptions are not valid, then you need to perform the **Non-Linear Static analysis**. For example, an object attached with a spring being applied under forces; refer to [Figure-2](#).

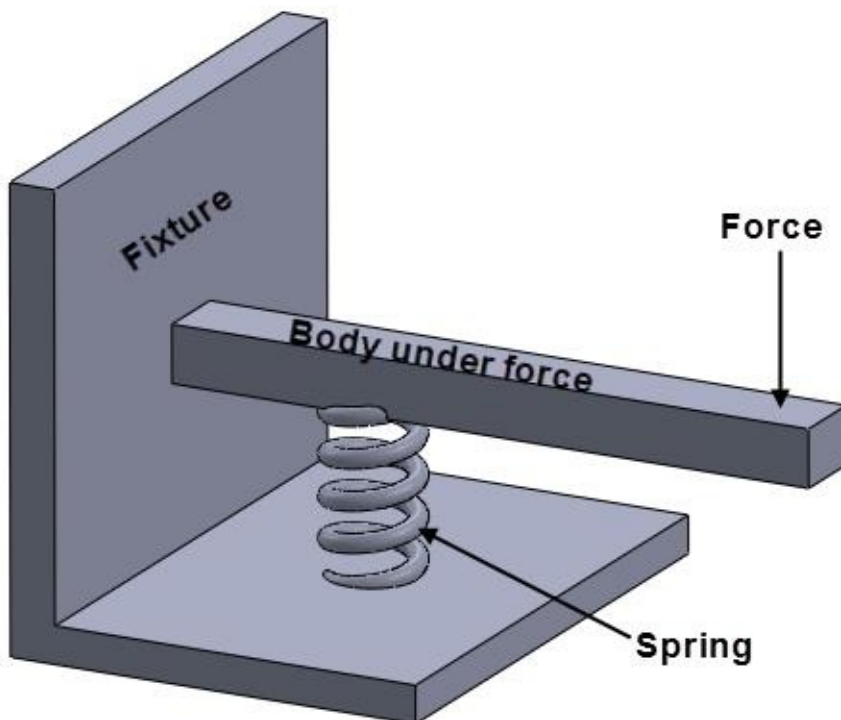


Figure-2. Non-linear static analysis example

Dynamic Analysis

In general, we have to perform dynamic analysis on a structure when the load applied to it varies with time. The most common case of dynamic analysis is the evaluation of responses of a building due to earthquake acceleration at its base. Every structure has a tendency to vibrate at certain frequencies, called natural 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 excited by a dynamic load that coincides with one of its natural frequencies, the structure undergoes large displacements. This phenomenon is known as ‘resonance’. Damping prevents the response of the structures to resonant loads. In reality, 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 first few modes of a model (those with the lowest natural frequencies), are normally important. The natural frequencies and corresponding mode shapes depend on the geometry of the structure, its material properties, as well as its support conditions and static loads. The computation of natural frequencies and mode shapes is known as modal analysis. When building the geometry of a model, you usually create it based on the original (undeformed) shape of the model. Some loading, like a structure’s self-weight, is always present and can cause considerable changes in the structure’s original geometry. These geometric changes may have, in some cases, significant impact on the structure’s modal properties. In many cases, this effect can be ignored because the induced deflections are small.

The following few topics – Random Vibration, Response Spectrum analysis, Time History analysis, Transient vibration analysis and Vibration modal analysis are extensions of dynamic analysis.

Random Vibration

Engineers use this type of analysis to find out how a device or structure responds to steady shaking of the kind you would feel riding in a truck, rail car, rocket (when the motor is on), and so on. Also, things that are riding in the vehicle, such as on-board electronics or cargo of any kind, may need Random Vibration Analysis. The vibration generated in vehicles from the motors, road conditions, etc. is a combination of a great many frequencies from a variety of sources and has a certain “random” nature. Random Vibration Analysis is used by mechanical engineers who design various kinds of transportation equipment.

Response Spectrum Analysis

Engineers use this type of analysis to find out how a device or structure responds to sudden forces or shocks. It is assumed that these shocks or forces occur at boundary points, which are normally fixed. An example would be a building, dam or nuclear reactor

when an earthquake strikes. During an earthquake, violent shaking occurs. This shaking transmits into the structure or device at the points where they are attached to the ground (boundary points).

Mechanical engineers who design components for nuclear power plants must use response spectrum analysis as well. Such components might include nuclear reactor parts, pumps, valves, piping, condensers, etc. When an engineer uses response spectrum analysis, he is looking for the maximum stresses or acceleration, velocity and displacements that occur after the shock. These in turn lead to maximum stresses.

Time History Analysis

This analysis plots response (displacements, velocities, accelerations, internal forces etc.) of the structure against time due to dynamic excitation applied on the structure.

Transient Vibration Analysis

When you strike a guitar string or a tuning fork, it goes from a state of inactivity into a vibration to make a musical tone. This tone seems loudest at first, then gradually dies out. Conditions are changing from the first moment the note is struck. When an electric motor is started up, it eventually reaches a steady state of operation. But to get there, it starts from zero RPM and passes through an infinite number of speeds until it attains the operating speed. Every time you rev the motor in your car, you are creating transient vibration. When things vibrate, internal stresses are created by the vibration. These stresses can be devastating if resonance occurs between a device producing vibration and a structure responding to. A bridge may vibrate in the wind or when cars and trucks go across it. Very complex vibration patterns can occur. Because things are constantly changing, engineers must know what the frequencies and stresses are at all moments in time. Sometimes transient vibrations are extremely violent and short-lived. Imagine a torpedo striking the side of a ship and exploding, or a car slamming into a concrete abutment or dropping a coffeepot on a hard floor.

Such vibrations are called “shock, “ which is just what you would imagine. In real life, shock is rarely a good thing and almost always unplanned. But shocks occur anyhow. Because of vibration, shock is always more devastating than if the same force were applied gradually.

Vibration Analysis (Modal Analysis)

By its very nature, vibration involves repetitive motion. Each occurrence of a complete motion sequence is called a “cycle.” Frequency is defined as so many cycles in a given time period. “Cycles per seconds” or “Hertz”. Individual parts have what engineers call “natural” frequencies. For example, a violin string at a certain tension will vibrate only at a set number of frequencies, which is why you can produce specific musical tones. There

is a base frequency in which the entire string is going back and forth in a simple bow shape.

Harmonics and overtones occur because individual sections of the string can vibrate independently within the larger vibration. These various shapes are called “modes”. The base frequency is said to vibrate in the first mode, and so on up the ladder. Each mode shape will have an associated frequency. Higher mode shapes have higher frequencies. The most disastrous kinds of consequences occur when a power-driven device such as a motor for example, produces a frequency at which an attached structure naturally vibrates. This event is called “resonance.” If sufficient power is applied, the attached structure will be destroyed. Note that ancient armies, which normally marched “in step,” were taken out of step when crossing bridges. Should the beat of the marching feet align with a natural frequency of the bridge, it could fall down. Engineers must design so that resonance does not occur during regular operation of machines. This is a major purpose of Modal Analysis. Ideally, the first mode has a frequency higher than any potential driving frequency. Frequently, resonance cannot be avoided, especially for short periods of time. For example, when a motor comes up to speed it produces a variety of frequencies. So it may pass through a resonant frequency.

Buckling Analysis

If you press down on an empty soft drink can with your hand, not much will seem to happen. If you put the can on the floor and gradually increase the force by stepping down on it with your foot, at some point it will suddenly squash. This sudden scrunching is known as “buckling.”

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.

In the normal use of most products, buckling can be catastrophic if it occurs. The failure is not one because of stress but geometric stability. Once the geometry of the part starts to deform, it can no longer support even a fraction of the force initially applied. The worst part about buckling for engineers is that buckling usually occurs at relatively low stress values for what the material can withstand. So they have to make a separate check to see if a product or part thereof is okay with respect to buckling.

Slender structures and structures with slender parts loaded in the axial direction buckle under relatively small axial loads. Such structures may fail in buckling while their stresses are far below critical levels. For such structures, the buckling load becomes a critical design factor. Stocky structures, on the other hand, require large loads to buckle, therefore buckling analysis is usually not required.

Buckling almost always involves compression; refer to [Figure-3](#). In mechanical engineering, designs involving thin parts in flexible structures like airplanes and automobiles are susceptible to buckling. Even though stress can be very low, buckling of local areas can cause the whole structure to collapse by a rapid series of ‘propagating buckling’. Buckling analysis calculates the smallest (critical) loading required buckling 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.

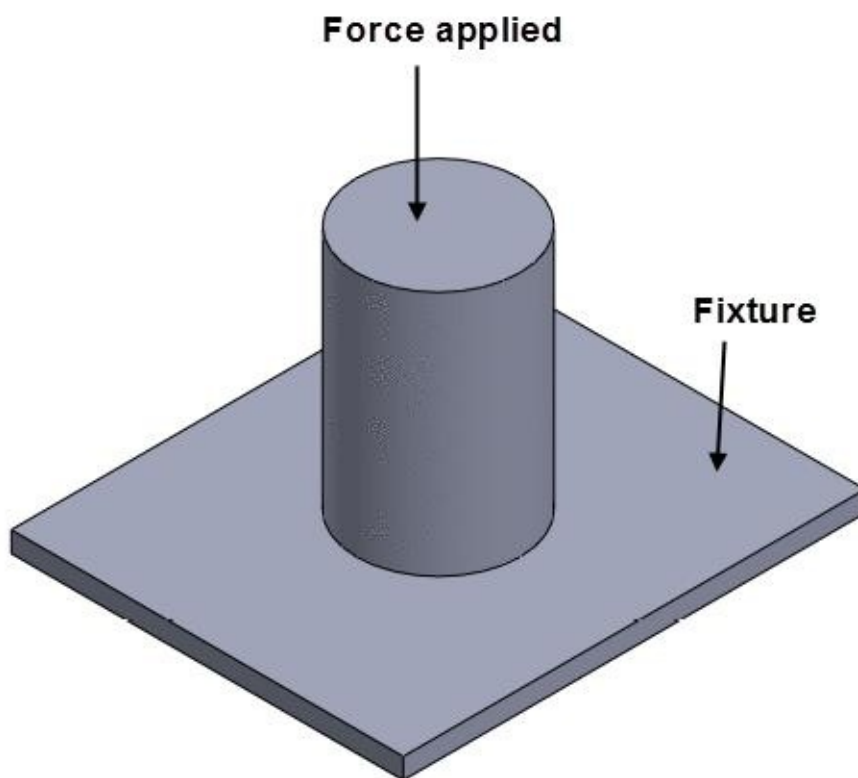


Figure-3. Buckling example

THERMAL ANALYSIS

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

There are two modes of heat transfer analysis.

Steady State Thermal Analysis

In this type of analysis, we are only interested in the thermal conditions of the body when it reaches thermal equilibrium, but we are not interested in the time it takes to reach this status. The temperature of each point in the model will remain unchanged until a change occurs in the system. At equilibrium, the thermal energy entering the system 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 at different instances 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 is interested in finding out the temperature of the fluid as a function of time. In addition to the thermal conductivity, we also need to specify density, specific heat, initial temperature profile, and the period of time for which solutions are desired.

Till this point, we have learned the basics of various analyses that can be performed in SolidWorks. Now, we will learn about the studies that can be performed in SolidWorks.

DROP TEST STUDIES

Drop test studies simulate the effect of dropping a part or an assembly on a rigid or flexible floor. To perform this study the floor is considered as planar and flat. The forces that are considered automatically for this study are gravity and impact reaction.

FATIGUE ANALYSIS

The fatigue is more over a study than analysis. But it is generally named as analysis. This analysis is used to check the effect of continuous loading and unloading of forces on a body. The base element for performing fatigue analysis are results of static, nonlinear, or time history linear dynamic studies.

PRESSURE VESSEL DESIGN STUDY

Pressure Vessel Design study allows you combine the results of static studies with the desired factors and interpret the results. The Pressure Vessel Design study combines the results of the static studies algebraically using a linear combination or the square root of the sum of the squares.

DESIGN STUDY

Design Study is used to perform an optimization of design. Using the Design Study you can:

- Define multiple variables using simulation parameters, or driving global variables.
- Define multiple constraints.
- Define multiple goals using sensors.
- Analyze models without simulation results. For example, you can minimize the mass of an assembly with the variables, density and model dimensions, the constraint, and volume.
- Evaluate design choices by defining a parameter that sets bodies to use different materials as a variable.

Till this point, you are familiar with the analyses that can be performed by using SolidWorks. But, Do you know how the software analyze the problems. The answer is FEA.

FEA

FEA, Finite Element Analysis, is a mathematical system to solve real-world engineering problems by simplifying them. In FEA by SolidWorks, the model is broken into small elements and nodes. Then, distributed forces are applied on each element and node. The cumulative result of forces is calculated and displayed in results. The elements in which a model can be broken into are given in [Figure-4](#), [Figure-5](#), and [Figure-6](#).

Common Finite Elements library for Linear Static and Dynamic Stress Analysis


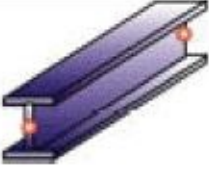
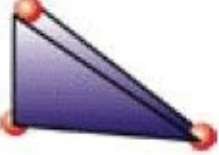

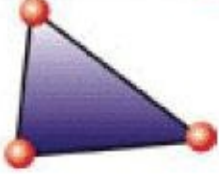


Element type	Illustration	Description
3-D Truss, 2-nodes		Truss elements are used to provide stiffness between two nodes. These elements transmit compressive and tensile loads along their axis. They do not carry any bending load.
3-D Beam, 2-nodes		Beam elements are used to provide elongational, flexural and rotational stiffness between two nodes. These elements can possess a wide variety of cross-sectional geometries including many standard types
3-D Membrane Plane Stress, 3-nodes		Membrane plane stress elements are used to model "fabric-like" structures, such as tents, cots, domed stadiums, etc. They support three translational degrees of freedom and in-plane (membrane) loading. Orthotropic material properties may be temperature dependent. Incompatible modes are available.
3-D Membrane Plane Stress, 4-nodes		Membrane plane stress elements are used to model "fabric-like" structures, such as tents, cots, domed stadiums, etc. They support three translational degrees of freedom and in-plane (membrane) loading. Orthotropic material properties may be temperature dependent. Incompatible modes are available.
2-D Elasticity, 3-nodes		Elasticity elements are used for plane strain, plane stress and axisymmetric formulations. They support two translational degrees of freedom. Orthotropic material properties may be temperature dependent. Incompatible modes are available.
2-D Elasticity, 4-nodes		Elasticity elements are used for plane strain, plane stress and axisymmetric formulations. They support two translational degrees of freedom. Orthotropic material properties may be temperature dependent. Incompatible modes are available.
3-D Brick, 4-nodes		Brick elements are used to simulate the behavior of solids. They support three translational degrees of freedom as well as incompatible displacement modes. Applications include solid objects, such as wheels, turbine blades, flanges, etc.

Figure-4. Finite elements list1

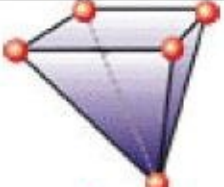
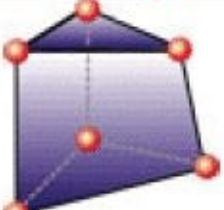
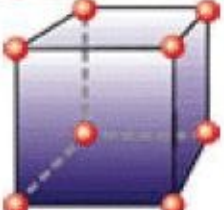
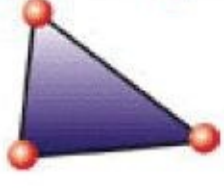


3-D Brick, 5-nodes		Same as above
3-D Brick, 6-nodes		Same as above
3-D Brick, 8-nodes		Same as above
3-D Plate, 3-nodes		<p>Plate elements are used in the design of pressure vessels, automobile body parts, etc. They support three translational and two rotational degrees of freedom as well as orthotropic material properties. An optional rotational stiffness around the perpendicular axis is automatically added to the node of each element.</p>
3-D Plate, 4-nodes		<p>A thin composite plate element is available for use in models such as mechanical equipment, bicycle frames, etc. A thick composite plate element is also available and can be used in models such as honeycomb sandwich structures, aerospace products, etc. Both thin and thick composite plate elements have no limitations regarding orientation or stacking sequence and support the Tsai-Wu, maximum stress and maximum strain failure criteria.</p>
Tetrahedral, 4-nodes		<p>Tetrahedral elements are used to model solid objects, such as gears, engine blocks and other unusually shaped objects. They support three translational degrees of freedom. They are also available in higher order formulations (mid-side nodes).</p>

Figure-5. Finite elements list2

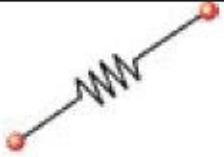

Boundary, 2-nodes		<p>Boundary elements are used in conjunction with other elements. A boundary element rigidly or elastically supports a model and enables the extraction of support reactions. Boundary elements are also used to impose a specified rotation or translation.</p>
Gap/Cable, 2-nodes		<p>The gap element simulates compression, where deflection makes two nodes touch and transmit force, such as when a ball bearing moves in a joint. Using gap elements, stresses, bending moments and axial forces where the bearing and joint meet can be determined.</p> <p>A cable element simulates tension, where two nodes moving away from each other a specified distance cause the element to become active. It is still a small-deflection, small-strain analysis, but with deflection-sensitive connectivity.</p>

Figure-6. Finite elements list3

We will be now use the information discussed above in SolidWorks Simulation. The process of starting SolidWorks Simulation is given next.

STARTING SOLIDWORKS SIMULATION

Before you try to start SolidWorks Simulation, make sure that you have installed it with SolidWorks and you fed the serial key for it during installation. Then, follow the steps given next.

- Start SolidWorks and open the model for which you want to perform the analyses.
- Click on the **SolidWorks Add-Ins** tab and select the **SolidWorks Simulation** button;

refer to [Figure-7](#).

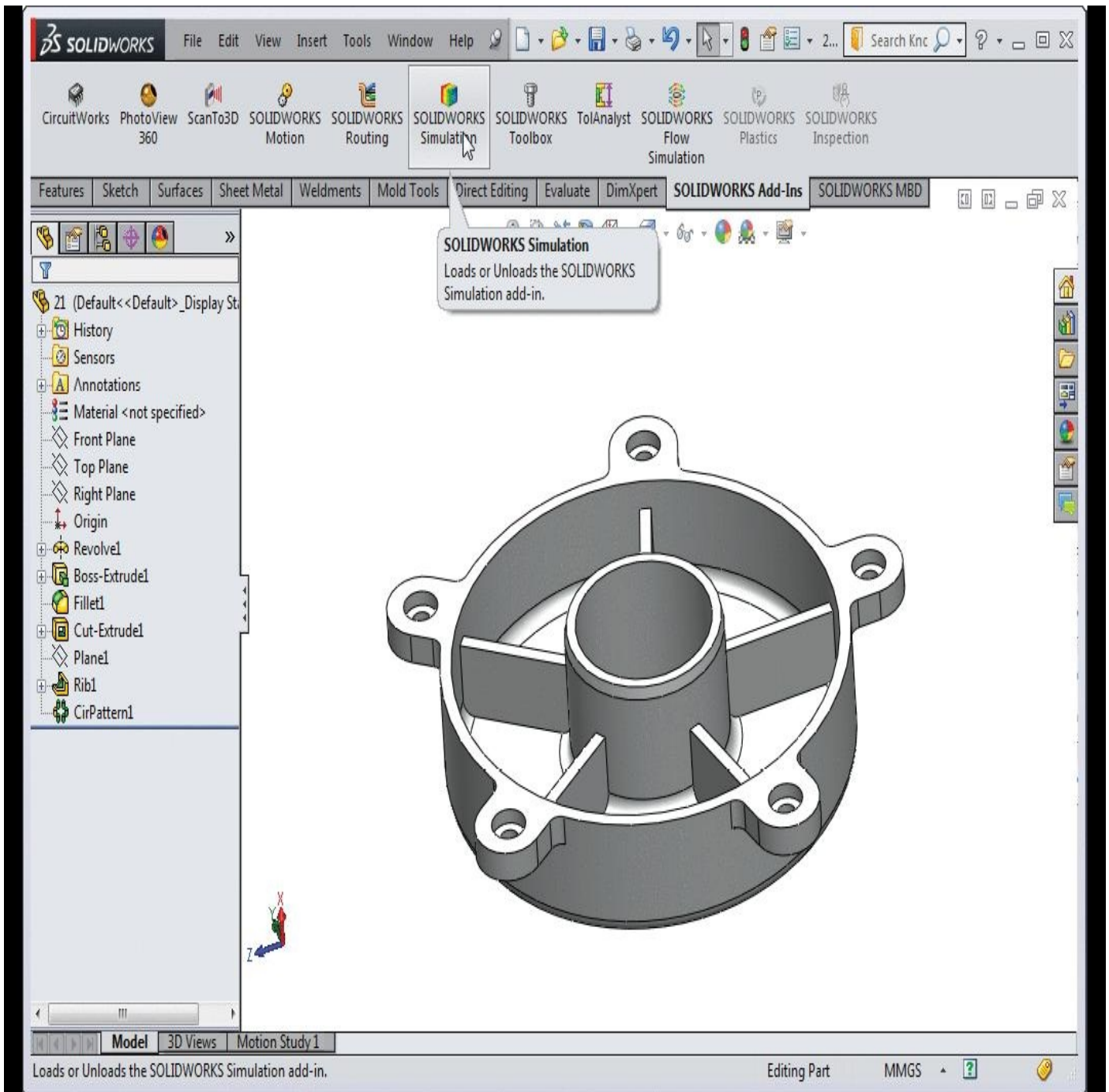


Figure-7. SolidWorks simulation button

- On clicking this button, **Simulation** tab will be added in the **Ribbon**.
- Click on the **Simulation** tab, the **Study Advisor** drop-down will be displayed.

There are two buttons available in this drop-down; **Study Advisor** and **New Study**. We will start with Study Advisor as it is good for novices to the software.

On clicking Study Advisor

- Click on the **Study Advisor** button from the drop-down. The **Simulation Advisor** pane will display in the right of the SolidWorks window; refer to [Figure-8](#).

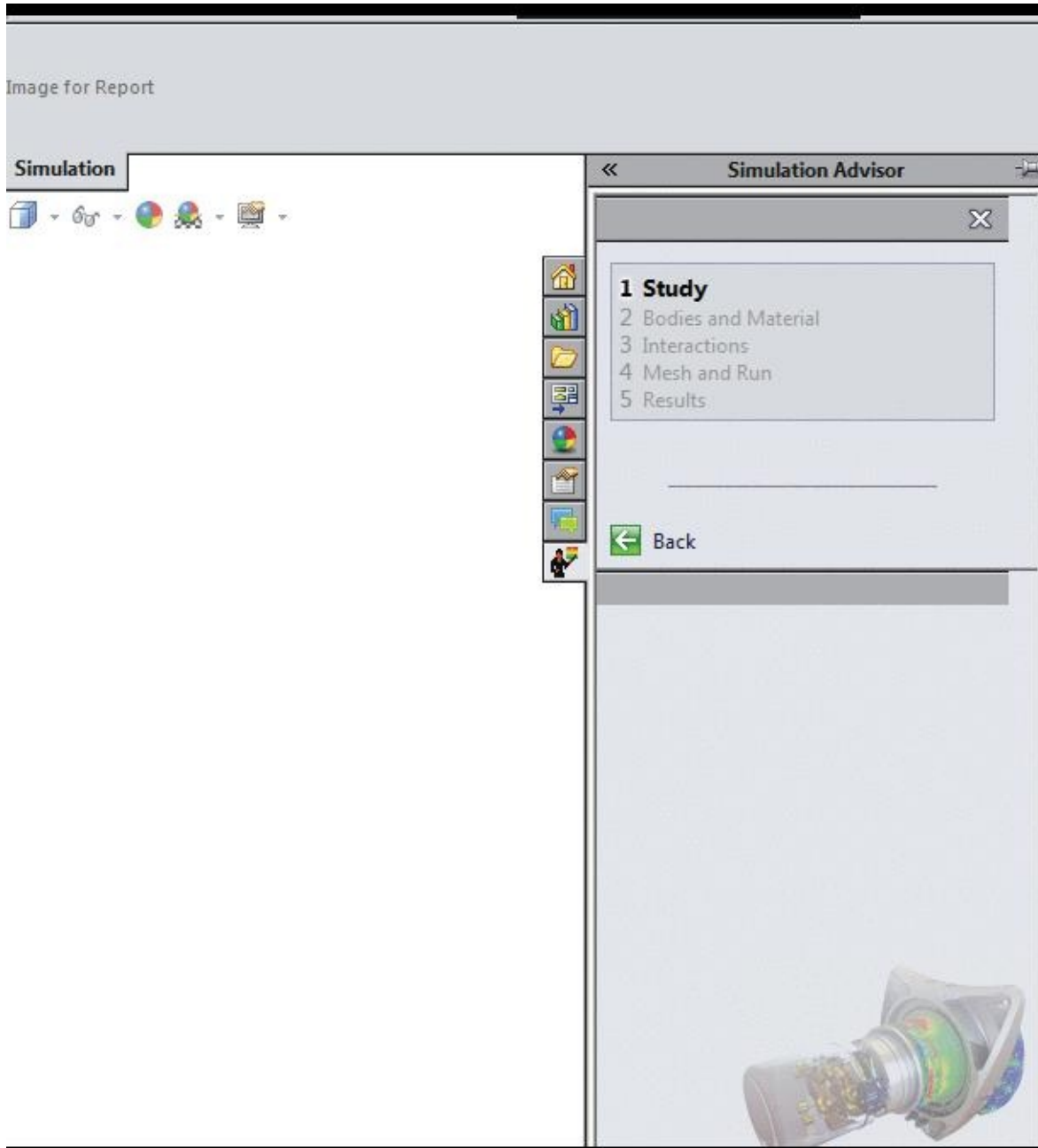


Figure-8. Simulation advisor pane

- Read the text in the Advisor and perform the actions accordingly. This is a guided process so it is recommended to perform some tricks on software by yourself. We will learn more about the advisor later in the book.

On clicking New Study

- Click on the **New Study** button from the drop-down. The **Study PropertyManager** will display; refer to [Figure-9](#).

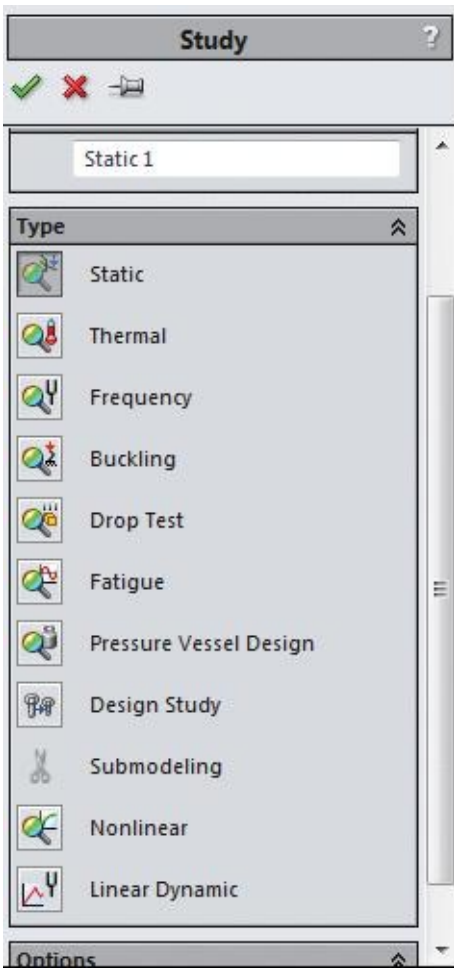


Figure-9. Study PropertyManager

The buttons available in the **Type** rollout of the **PropertyManager** refer to different type of analyses.

We will discuss these analyses one by one in the upcoming chapters.

BASICS OF ANALYSES

CHAPTER 2



Topics Covered

The major topics covered in this chapter are:

- *Starting Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

STARTING ANALYSIS

There are two methods to start a analysis in SolidWorks Simulation; using **Study Advisor**

and using **New Study** button. We will use the **New Study** button throughout the book. Although we will give short notes for using **Study Advisor** to perform the analyses. The steps to start static analysis are given next.

- Click on the **New Study** button from the **Study Advisor** drop-down in the **Simulation** tab of **Ribbon**. The **Study PropertyManager** will display.
- Click on the **Static** button from the **PropertyManager** and specify the name of analysis as desired in the **Name** rollout.
- Click on the **OK** button from the **PropertyManager**. The interface will display as shown in [Figure-1](#).

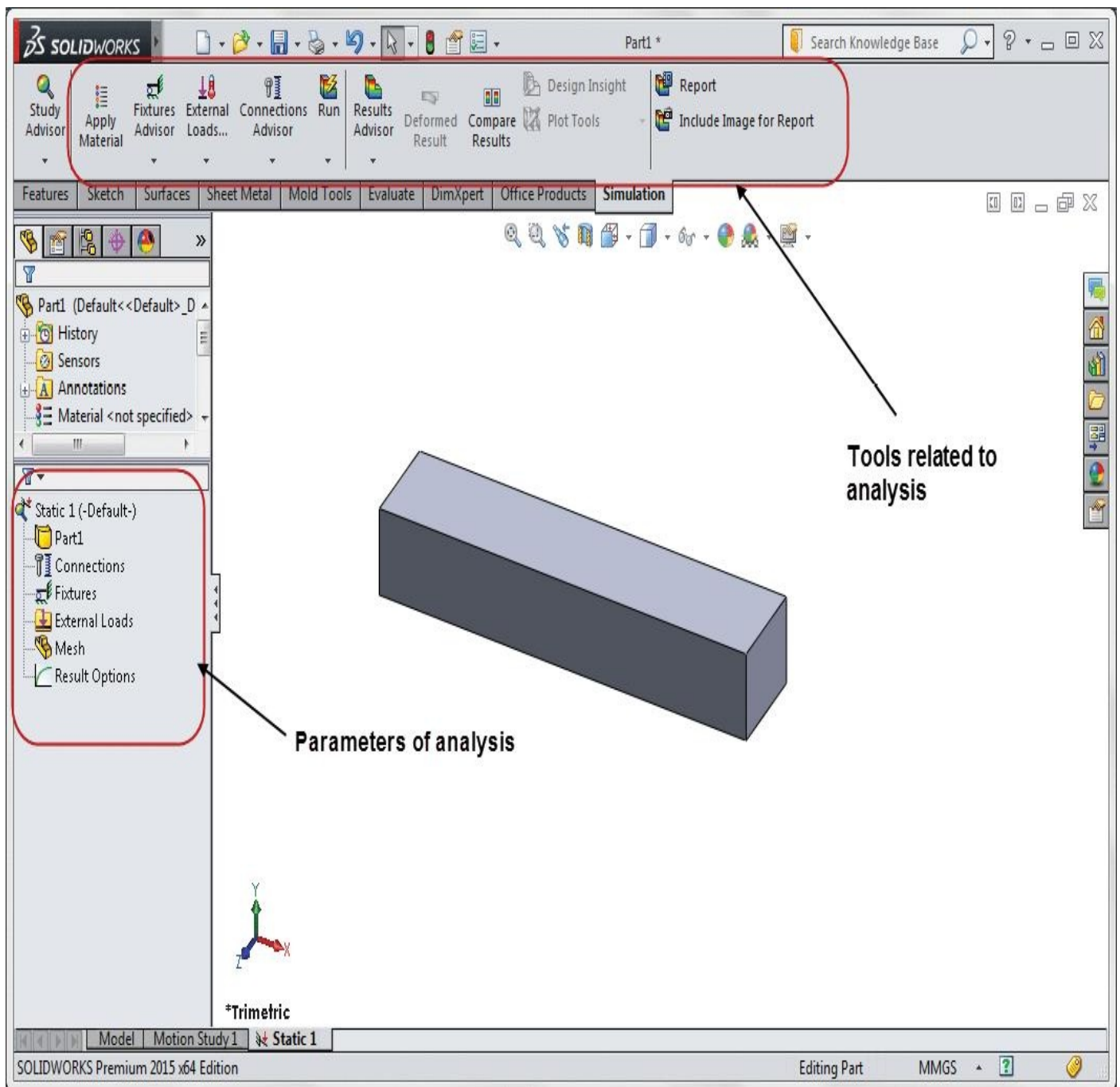


Figure-1. Static analysis interface

The first step for this analysis is to apply material on the model.

APPLYING MATERIAL

Material is a key input for any analysis. The result of analysis is directly related to material of the object. There are a few properties of material like, ultimate strength, hardness, and young's modulus which play important role in success/failure of the object under specified load. Also, material determines the application of object in real world. For example, we do not use glass to make pistons in engine. The steps to apply material on object are given next.

- Click on the **Apply Material** button from the **Simulation** tab in the **Ribbon**. The **Material** dialog box will display as shown in [Figure-2](#).

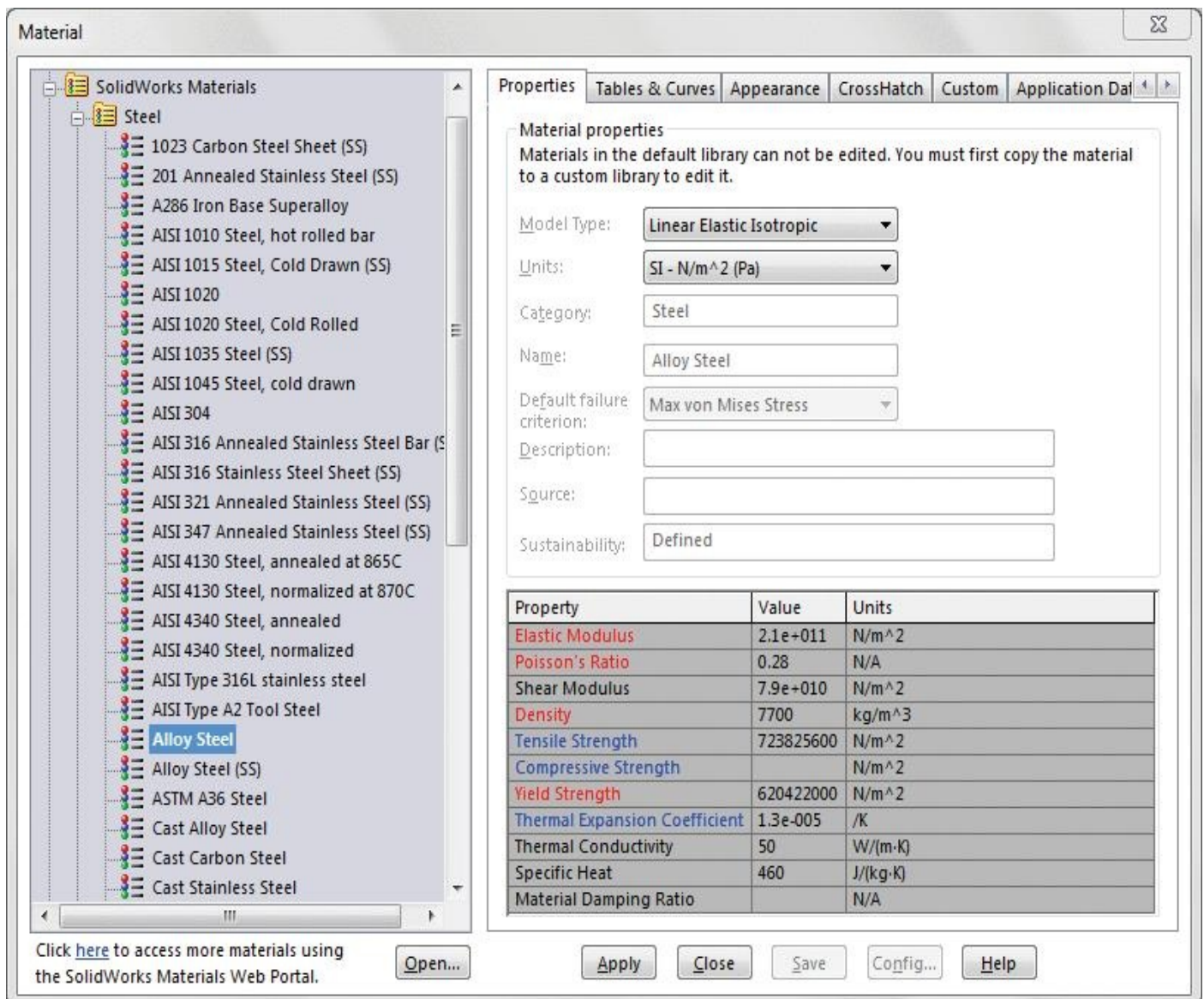


Figure-2. Material dialog box

A library of standard materials is available in this box for selection.

- Select the desired material from left pane of the box. The related parameters will be displayed at the right.
- Click on the **Apply** button to apply the material.
- Click on the **Close** button to close the box.

Although, we have a big library of standard materials available but still there is always a need to custom material. Now, we will discuss about adding custom materials in library.

Adding Custom Materials

Big researches are going on to enhance the properties of engineering materials to make them lighter and stronger. During your job, you may come across a material whose properties are not available in the library. In such a case, follow the steps given next to create custom material.

- Right-click on the **Material** option from the **Feature Manager Design Tree** and select the **Edit Material** option from the short-cut menu displayed; refer to [Figure-3](#). The Material dialog box will be displayed as shown in [Figure-2](#) earlier.

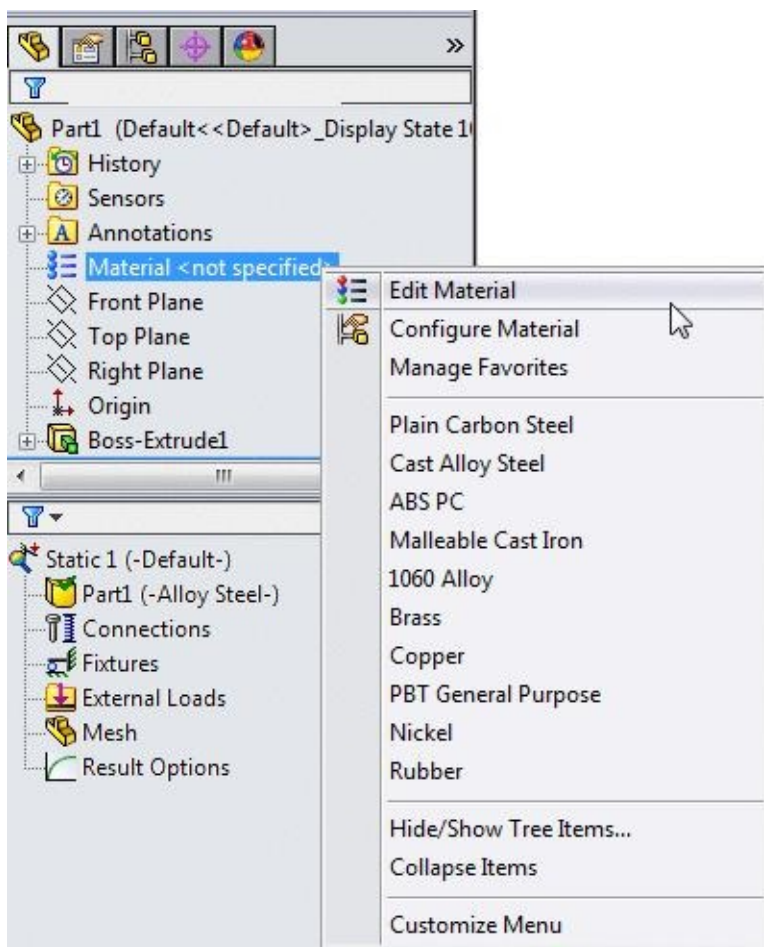


Figure-3. Edit Material option

- Right-click on the material whose properties are close to the material you want to create and select the **Copy** option from the short-cut menu displayed; refer to [Figure-4](#).

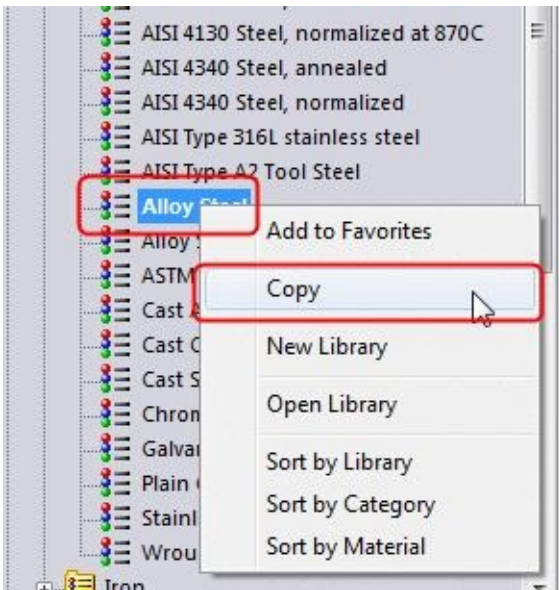


Figure-4. Copy option for material

- Now, move to the bottom of material list and expand the **Custom Material** category; refer to [Figure-5](#).



Figure-5. Custom Material category

- Right-click on the **Custom Materials** category and select the **New Category** option to add a new sub-category in it. Specify the desired name; refer to [Figure-6](#).

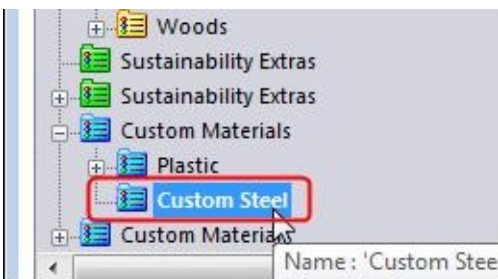


Figure-6. New category added

- Right-click on the new sub-category and select the **Paste** option from the shortcut menu;

refer to [Figure-7](#). The copied material will be added in the sub-category but with the edit-able features. Note that you can start with a completely new material by using the **New Material** option in place of **Paste** option.

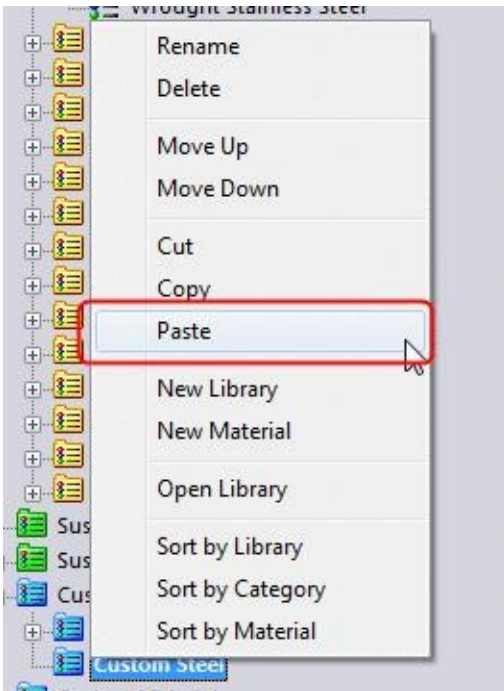


Figure-7. Paste option for materials

- Click on the added material, the options related to the material will be displayed on the right in dialog box; refer to [Figure-8](#).
- Click in the edit boxes and specify the desired values.
- Click on the **Save** button to save the parameters and then click on the **Close** button. Now, you can use this custom material like standard materials.

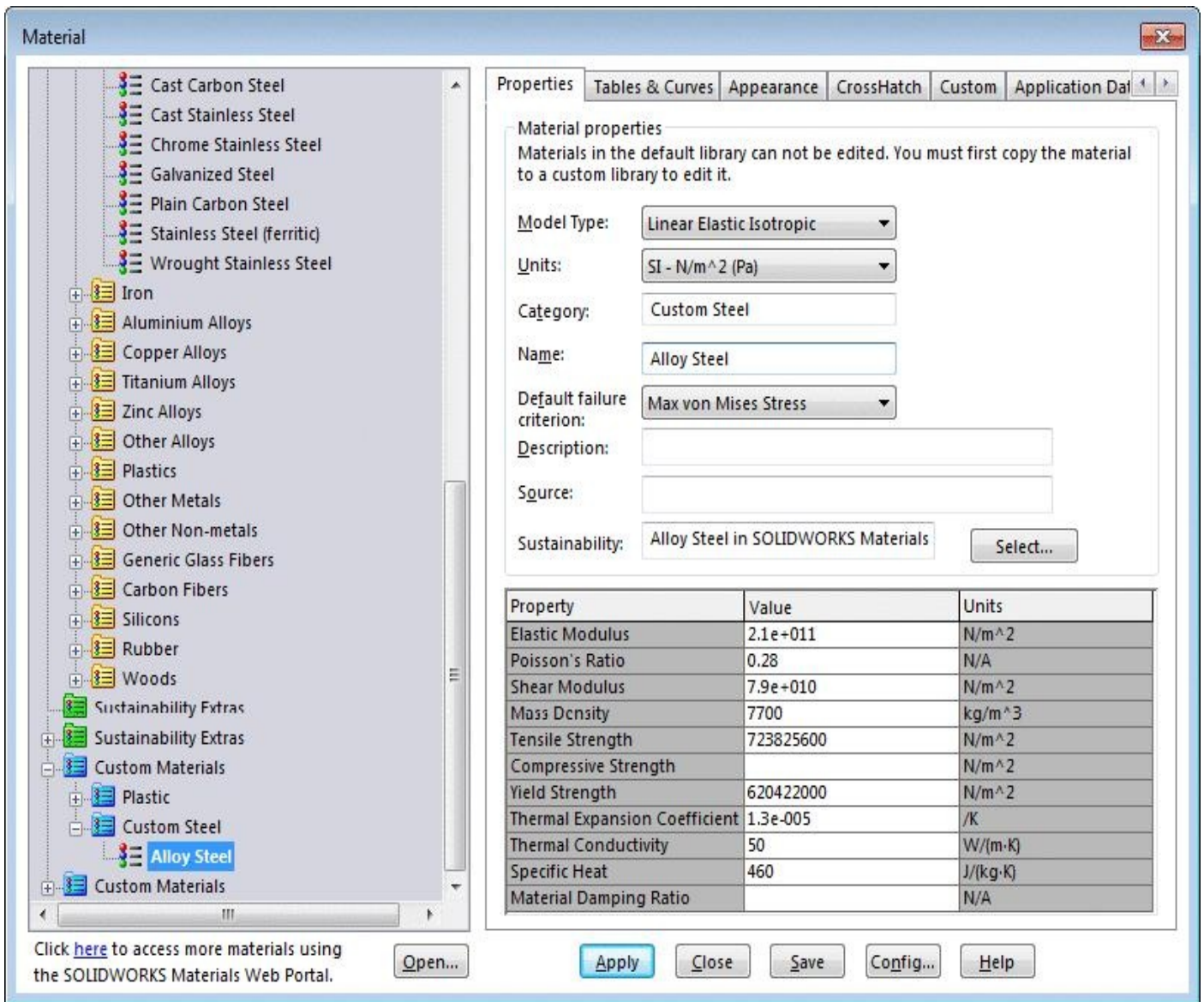


Figure-8. Details of custom material

DEFINING FIXTURES

To perform any analysis, it is important to fix the object by some constrains so that the effect of applied forces can be checked. If you do not apply fixtures then the object will be free to move in the direction of forces applied and no stress or strain will be induced in it. There are various options for fixing the object. All these options are explained next.

Fixtures Advisor

This button is used to create fixtures in a guided way. An interactive pane is used to apply fixtures on the basis of your responses. The steps to use this tool are given next.

- Click on the **Fixtures Advisor** button from the **Fixtures Advisor** drop-down in the

Ribbon. The **Simulation Advisor** will display in the right of the window; refer to [Figure-9](#).

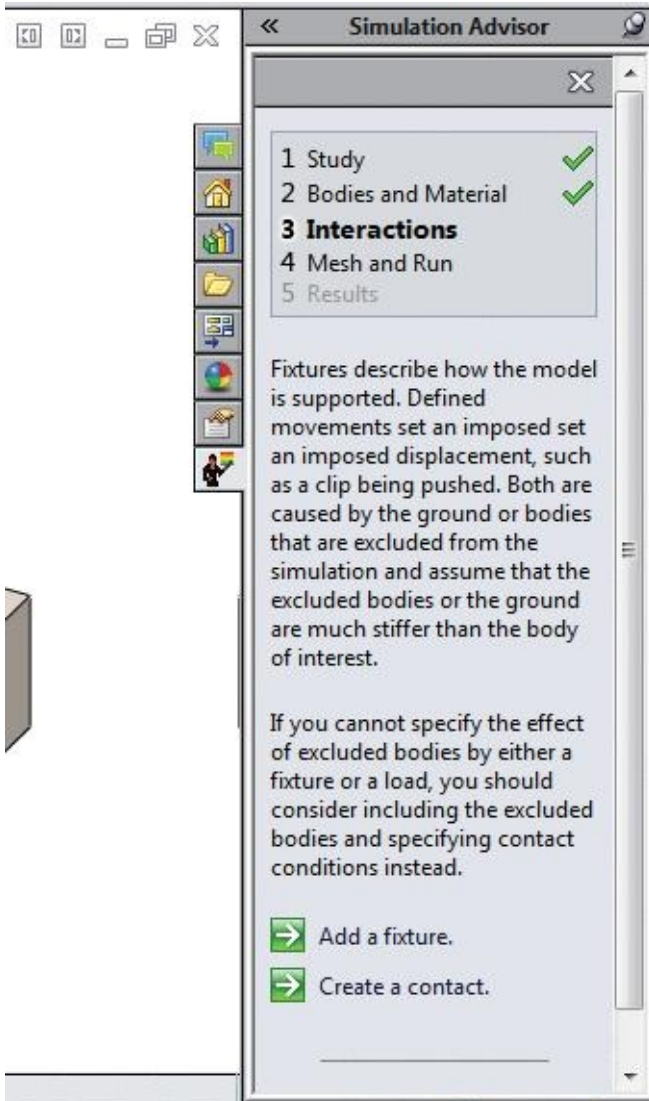


Figure-9. Simulation Advisor

- Click on the **Add a fixture** option from the **Simulation Advisor** pane. The **Fixture PropertyManager** will be displayed; refer to [Figure-10](#).

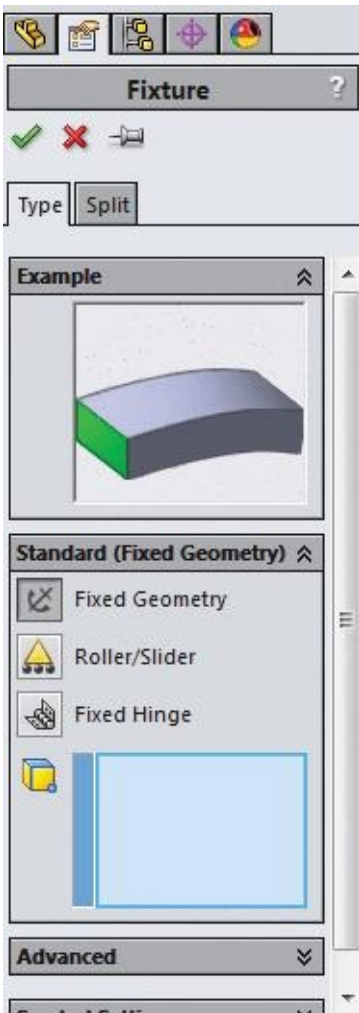


Figure-10. Fixture PropertyManager

Note that the options displayed in the Property Manager can be directly chosen from the **Fixture Advisor** drop-down in the **Ribbon**; refer to [Figure-11](#). Various options in the **PropertyManager** and the drop-down are discussed next.

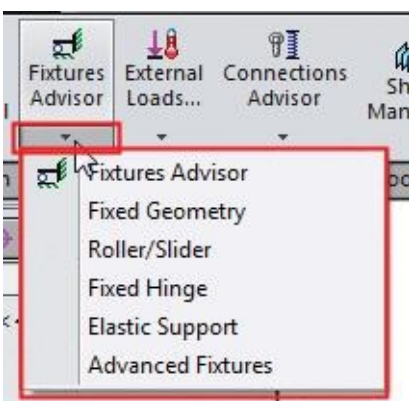


Figure-11. Fixtures Advisor drop-down

Fixed Geometry

This is the most commonly used fixture type. Using this fixture type, you can fix the face,

edge or point of the object. To apply this fixture, follow the steps given next.

- Click on the **Fixed Geometry** button from the **Fixtures Advisor** drop-down. The **Fixture PropertyManager** will display as shown in [Figure-10](#).
- Make sure that the **Fixed Geometry** button is selected in the **Standard (Fixed Geometry)** rollout.
- Select the face that you want to be fixed; refer to [Figure-12](#).

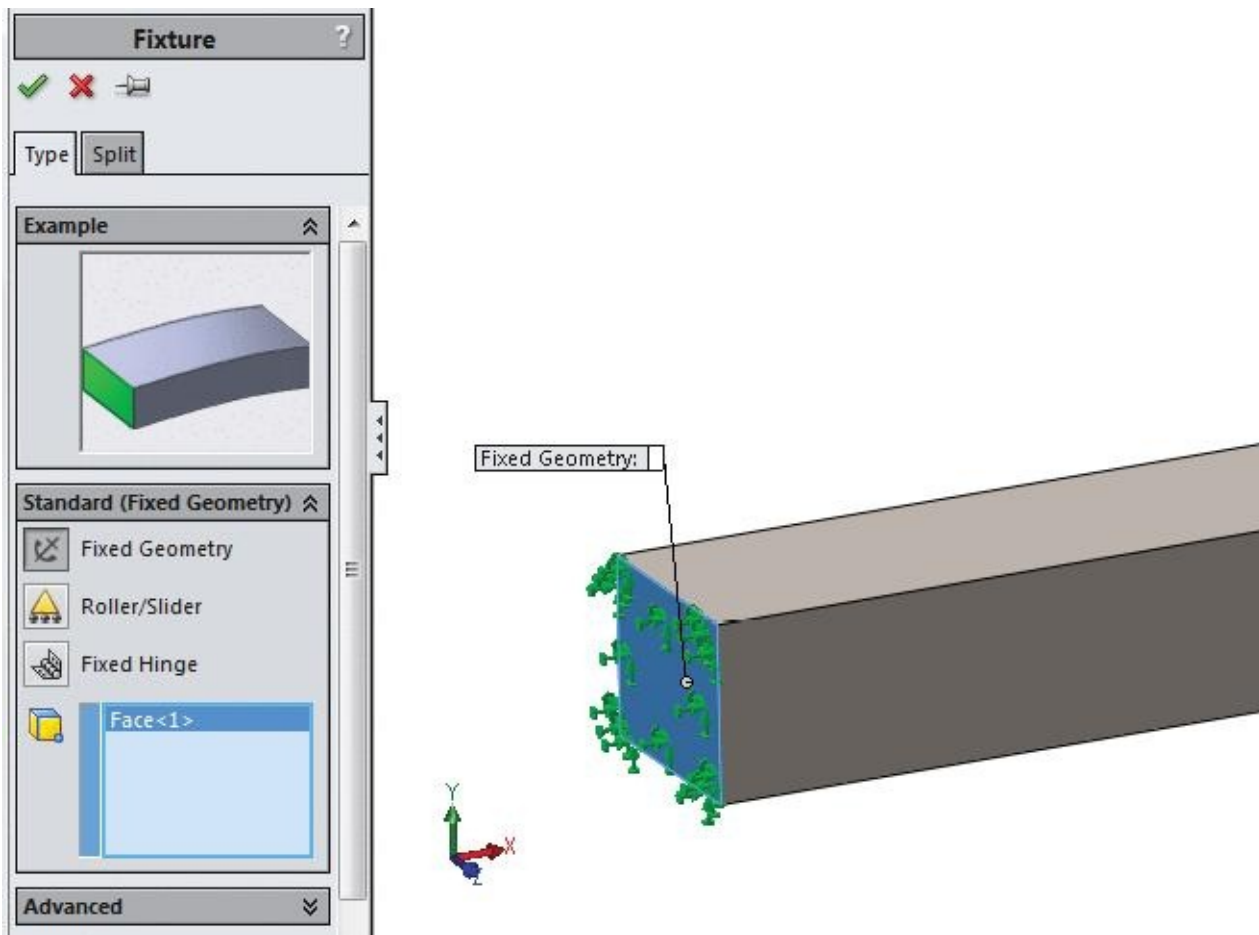


Figure-12. Face selected for fixing

- Click **OK** button to define the fixture.

You can also split a face before click **OK** to precise the fixture. The steps to do so are given next.

- Click on the **Split** tab from the **Fixture PropertyManager**. The **PropertyManager** will display as shown in [Figure-13](#).
- Click on the **Create Sketch** button from the **PropertyManager**. You are asked to select a plane/face for sketching.

- Select the desired plane to sketch the splitting line.

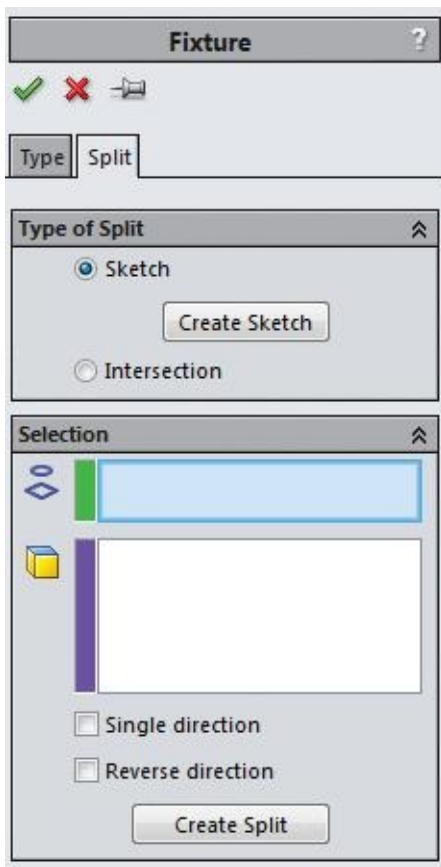


Figure-13. Fixture PropertyManager with split tab

- Create the splitting line by using the **Line** tool; refer to [Figure-14](#).
- Click on the **Exit Sketch** button displayed in **Split** box in viewport. You are asked to select a face.
- Select the faces that you want to split by the projection of drawn sketch line; refer to [Figure-15](#).

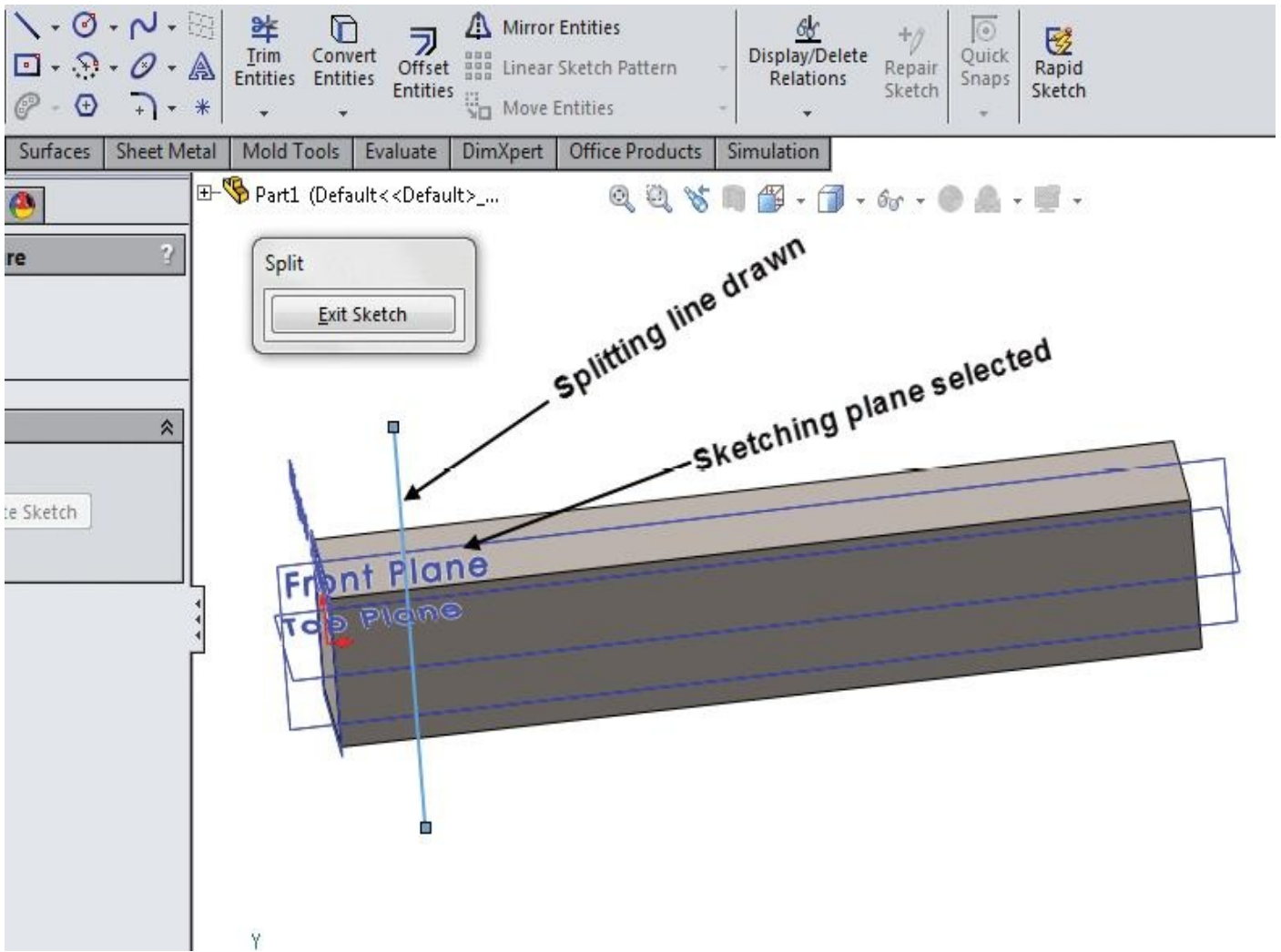


Figure-14. Splitting line drawn

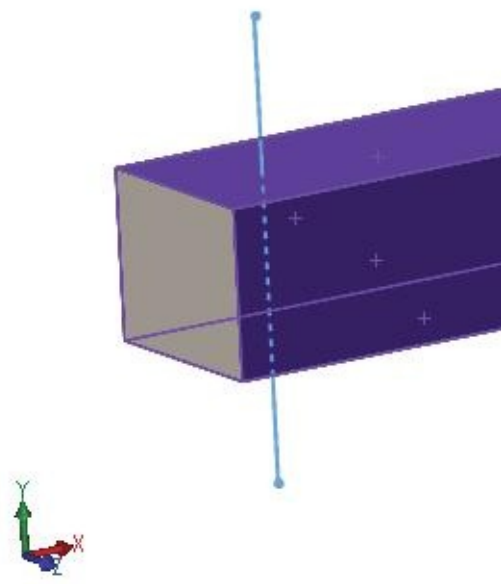
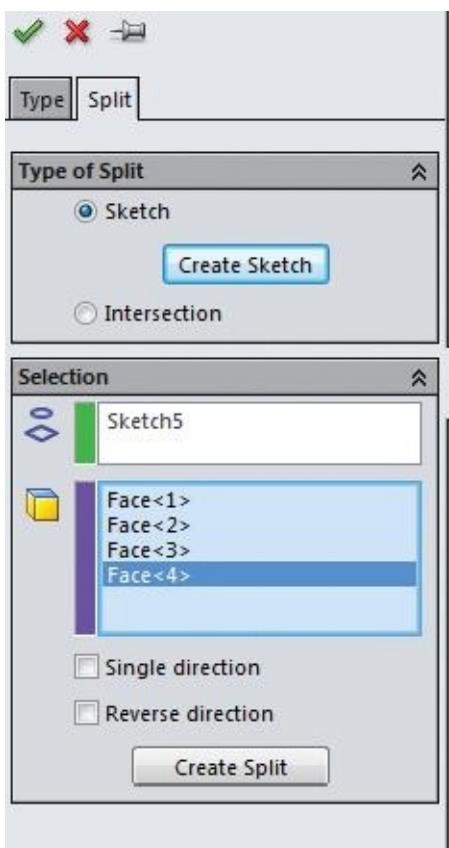


Figure-15. Faces selected for splitting

- Click on the **Create Split** button from the **Selection** rollout of the **PropertyManager**. The selected faces will get split; refer to [Figure-16](#).

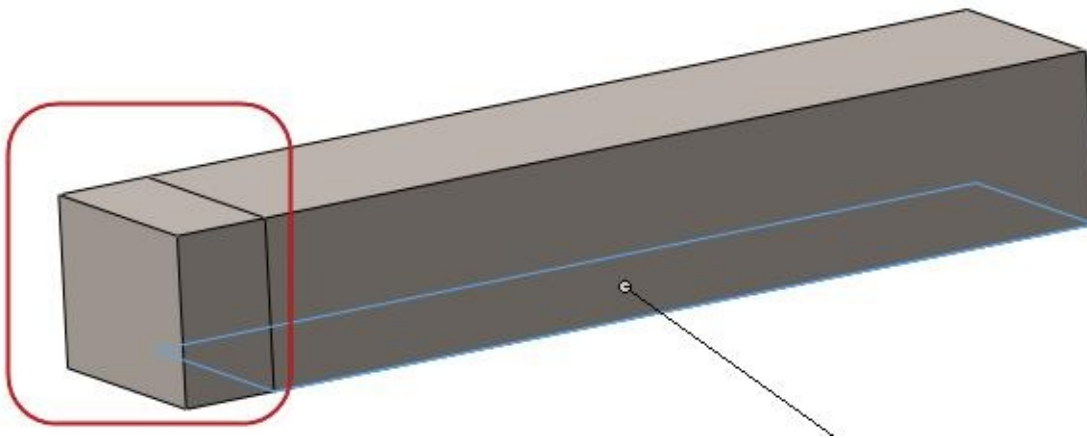


Figure-16. Faces after splitting

- Click on the **Type** tab of the **PropertyManager** to display options related to fixture.
- Click on the faces that you want to fix and remove unwanted faces by right-clicking and then selecting the **Delete** option from the shortcut menu; refer to [Figure-17](#).

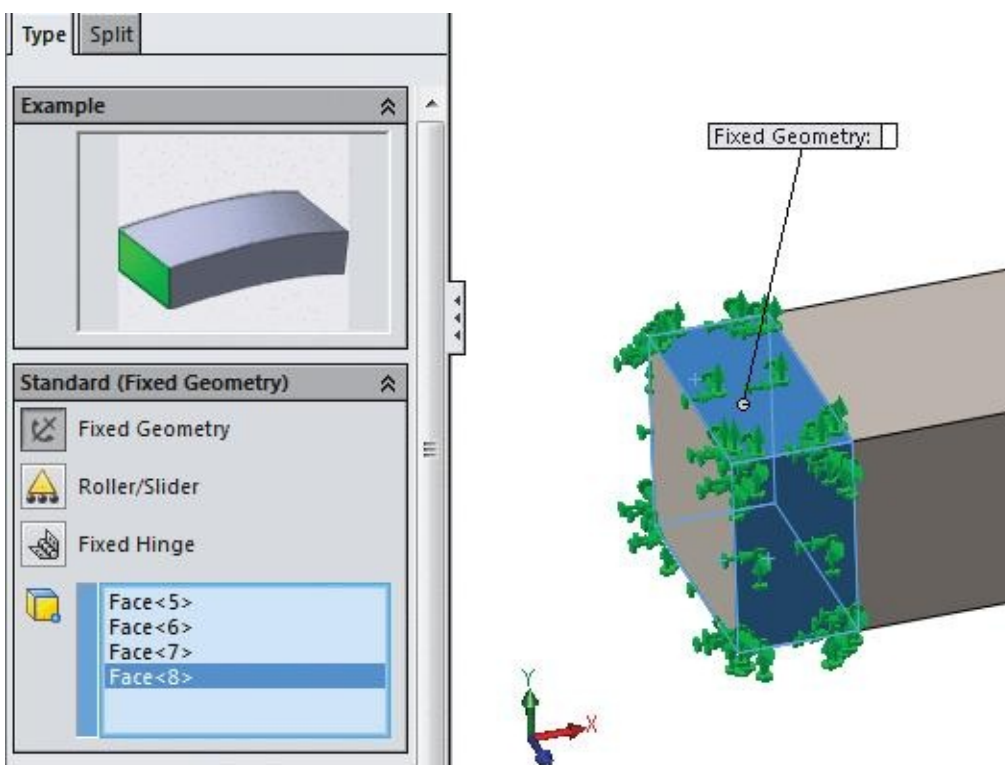


Figure-17. Splitted faces selected for fixing

- Click on **OK** button from the **PropertyManager** to fix the faces.



Where should we use the Fixed Geometry fixture? The answer is: we use this constraint where we know that the selected face is lying on a hard surface and it cannot move due to action of any applied forces. A live example can be a building fixed to earth by its base. We expect the building not to move automatically if load increases due to some unexpected guests.

Roller/Slider

To option is used to fix a face in such a way that it can slide/roll in its plane but cannot move perpendicular to the plane. The steps to fix a face by using this option are given next.

- Click on the **Roller/Slider** button from the **Fixtures Advisor** drop-down. The **Fixture PropertyManager** will be displayed similar to the one discussed earlier but with the **Roller/Slider** button selected; refer to [Figure-18](#).

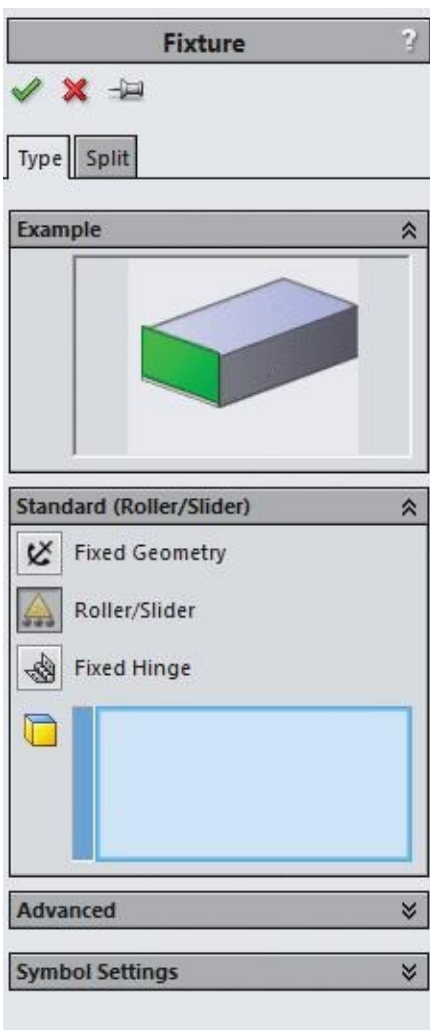


Figure-18. Fixture PropertyManager for roller slider fixture

- Select the face that you want to allow for sliding or rolling. Refer to [Figure-19](#).
- Click on the **OK** button from the **PropertyManager** to apply the fixture.

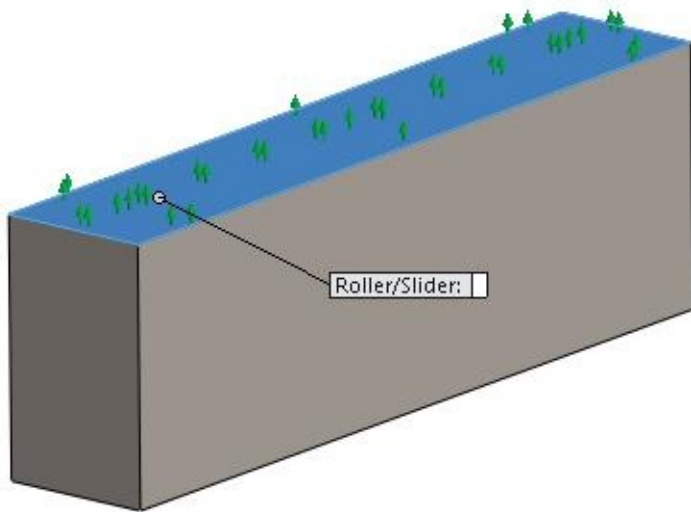


Figure-19. Slider face selection



Here the question comes, **where should we use the Roller/Slider fixture?** The answer is: we use this constraint **where object can roll or slide on a surface but can not move normal to it**. A live example can be Tyre of an automobile which rolls on the road and it is expected not to move up towards the sky or down.

Fixed Hinge

This option is used to fix a cylindrical face in such a way that the face react to be hinged. The steps to apply hinge fixture are given next.

- Click on the **Fixed Hinge** tool from the **Fixtures Advisor** drop-down. The **Fixture PropertyManager** will be displayed with the **Fixed Hinge** button selected; refer to [Figure-20](#).
- Select a cylindrical face that you want to fix as hinge; refer to [Figure-21](#).
- Click on the **OK** button to apply the fixture.

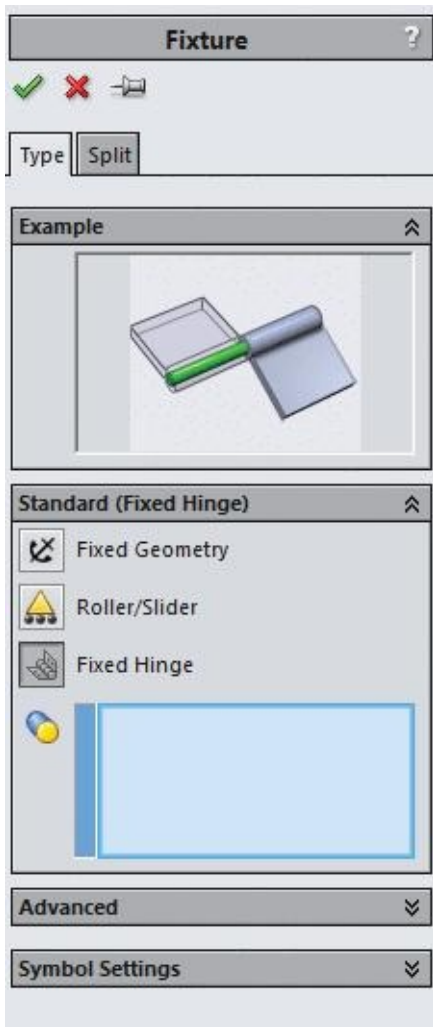


Figure-20. Fixture PropertyManager for fixed hinge fixture

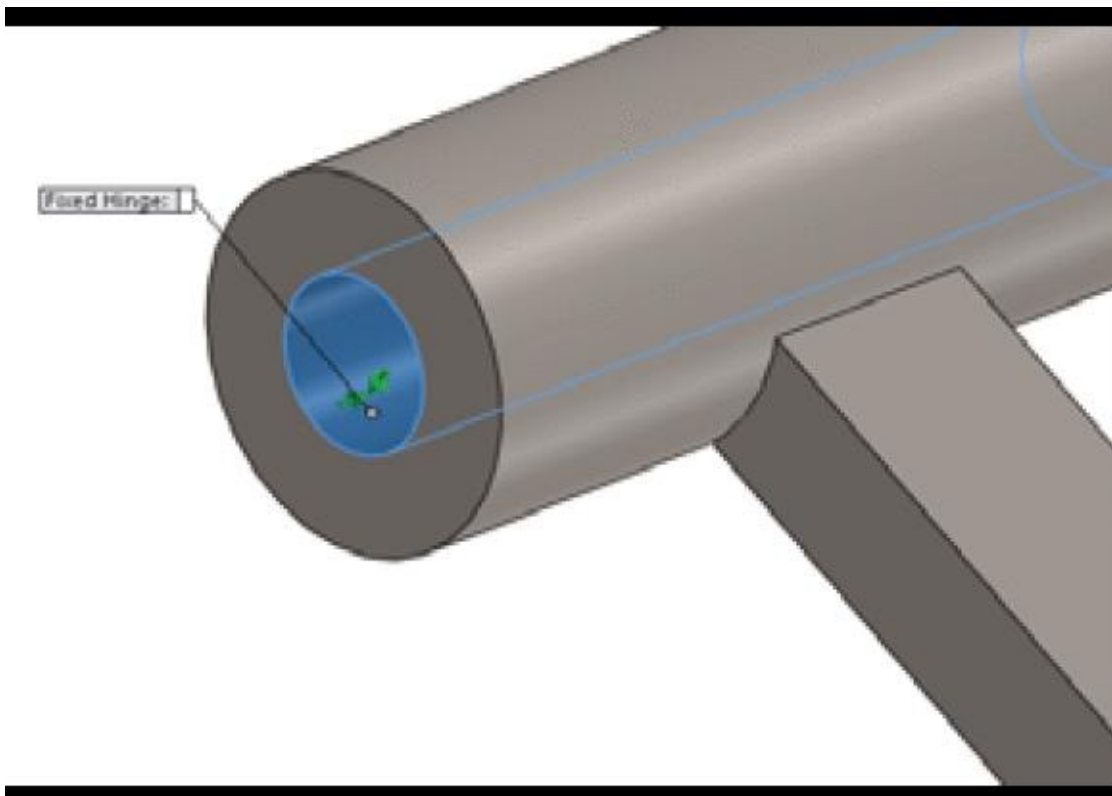


Figure-21. Face selected for hinge



Where should we use the Fixed Hinge fixture? The answer is: we use this constraint **where object can revolve around the axis of another**. A live example can be car hood (also call car bonnet in some part of world).

Elastic Support

This option is used to insert an elastic support between floor and object. This type of fixture is generally applied to rubber or plastic parts. The steps to apply this fixture are given next.

- Click on the **Elastic Support** button from the **Fixtures Advisor** drop-down. The **Connector PropertyManager** will display as shown in [Figure-22](#).

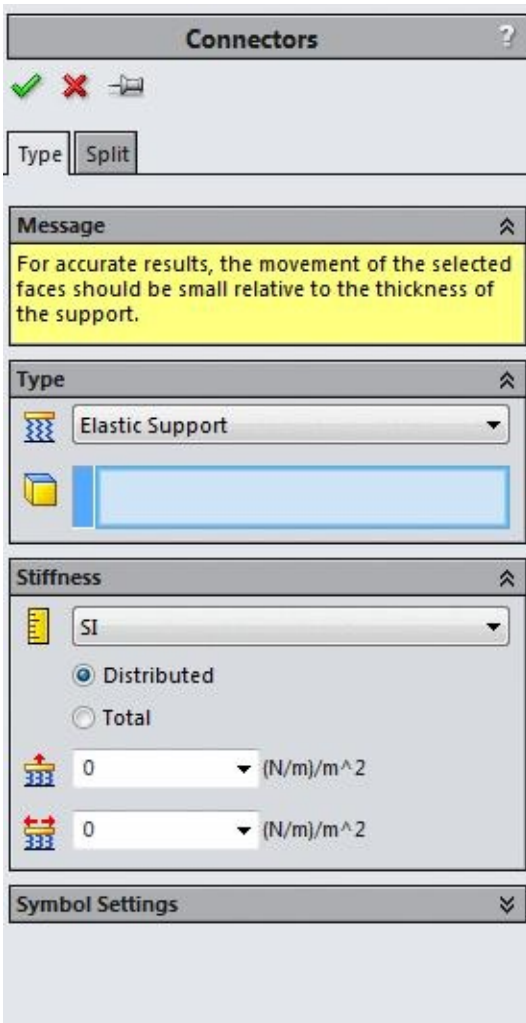


Figure-22. Connectors PropertyManager

- Select the face that you want to make as elastic support; refer to [Figure-23](#).

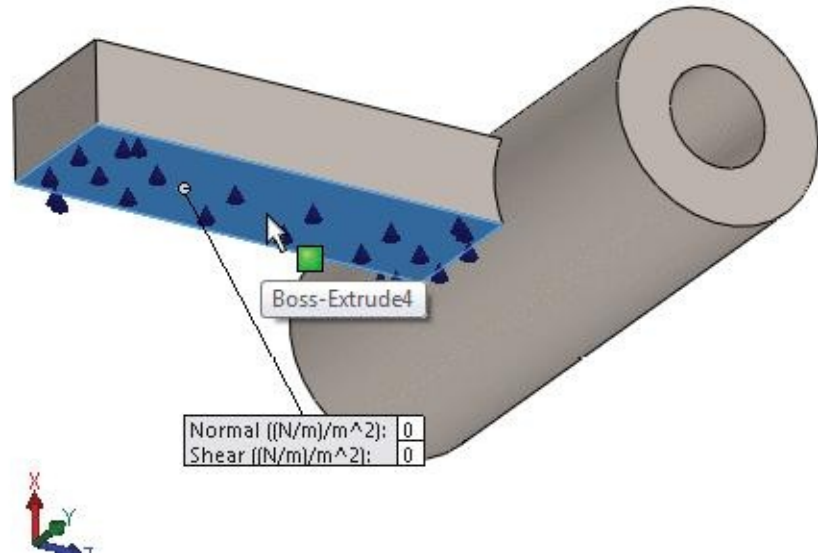
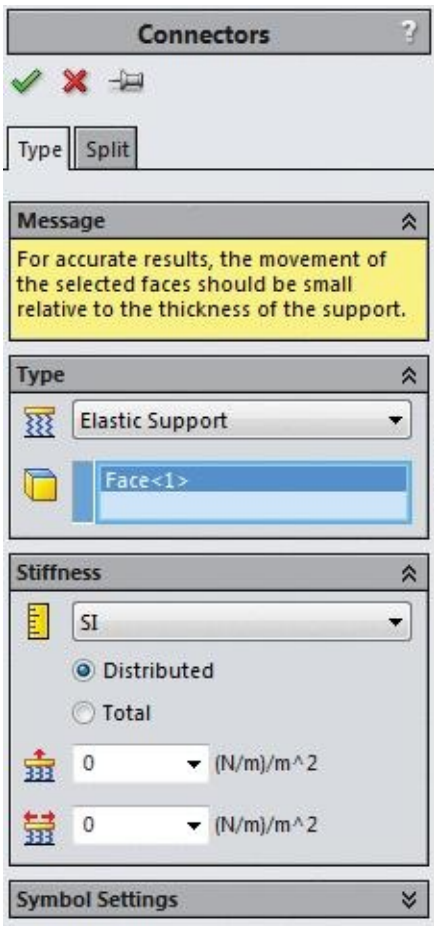


Figure-23. Selecting face for elastic support

- Click in the drop-down of **Stiffness** rollout to specify the type of units to be used to specify the value of stiffness.
- Select the desired option from the **SI**, **English (IPS)**, and **Metric (G)** option.
- Click in the **Normal** edit box and specify the stiffness of elastic support in normal direction to the selected face.
- Click in the **Shear** edit box and specify the value of stiffness in the lateral direction (also known as stiffness in shear direction).
- You can specify the total stiffness in normal and shear direction by selecting the **Total** radio button in place of **Distributed** in the Stiffness rollout.
- Click on the **OK** button to define the fixture.



Where should we use the Elastic Support fixture? The answer is: we use this constraint **where object is placed or fastened on a soft surface like wood or rubber**. A live example can be an object placed on wood, rubber or any spring base. I hope you have seen kids jumping on the bed which is a kind of elastic support.

Advanced Fixtures

There are some advanced fixtures available in SolidWorks Simulation for constraining the objects. To apply these fixtures follow the steps given next.

- Click on the **Advanced Fixtures** button from the **Fixtures Advisor** drop-down in the **Ribbon**. The **Fixture PropertyManager** will be displayed as shown in [Figure-24](#).

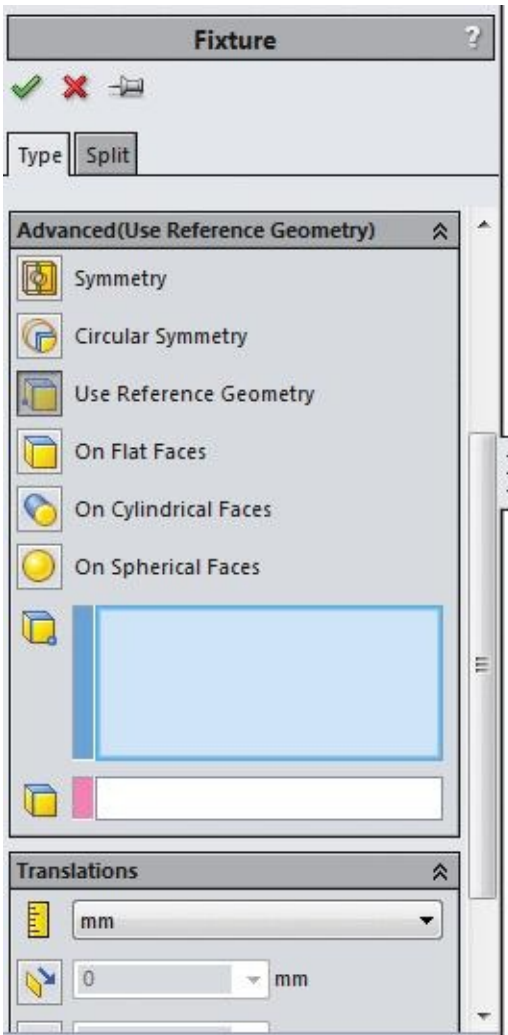


Figure-24. Advanced fixture options

The options available in this **PropertyManager** are discussed next.

Symmetry

This button is used to simplify the geometry of the model being analyzed. This option gives us freedom to analyze half of the body and hence reducing size of the problem. Based on this result the effect on whole body can be analyzed. This option is not recommended if you are performing Buckling or frequency studies. The steps to use this button are given next.

- Click on the **Symmetry** button from the **PropertyManager**. You are asked to select a face to define symmetry.
- Select the face that is symmetric half or quarter of the main part. Refer to [Figure-25](#). Note that the model displayed in the figure is having a center line in its sketch and about it the object is mirrored.

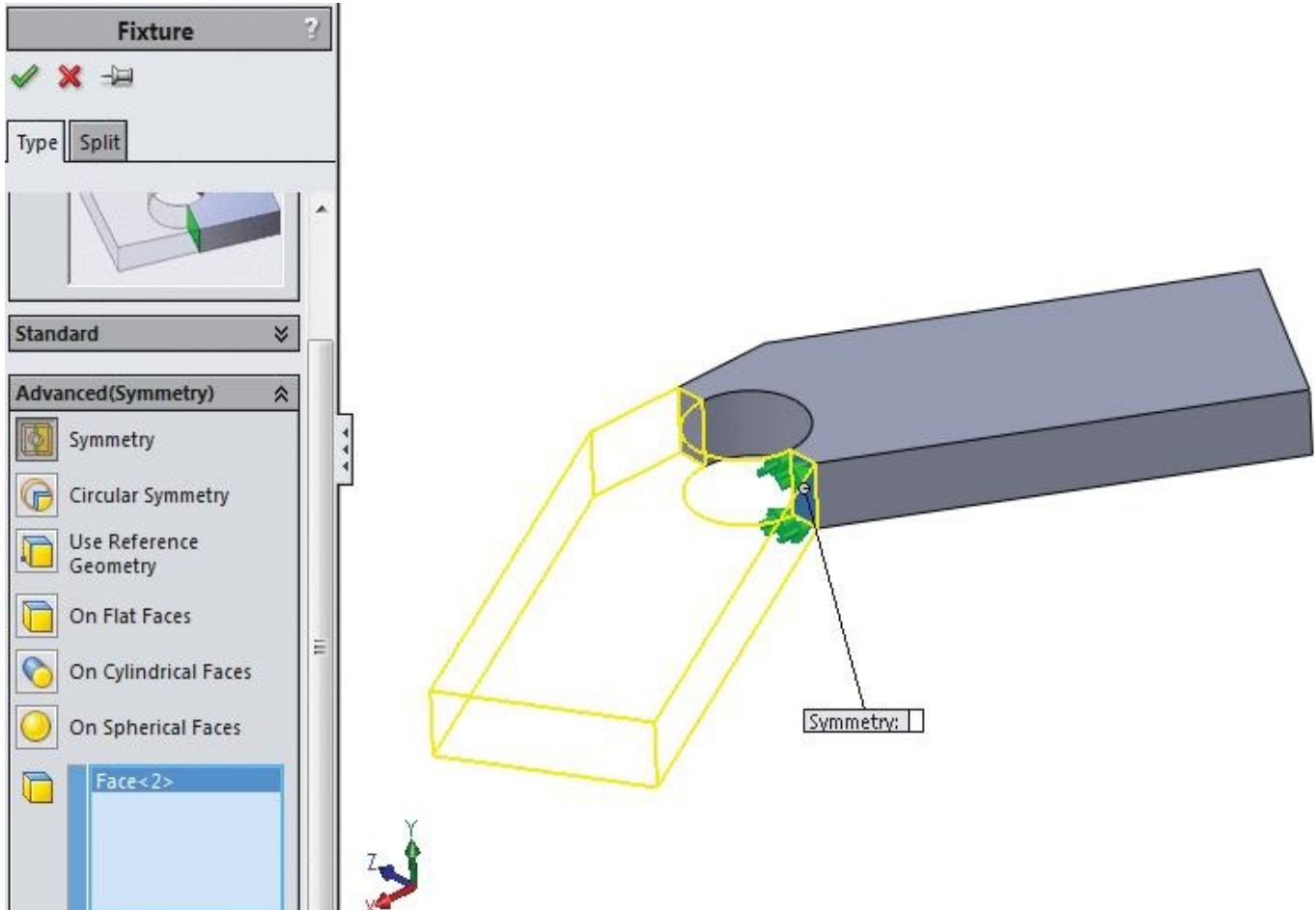


Figure-25. Symmetric fixture application

- Click on the **OK** button from the **PropertyManager** to apply the fixture.



Where should we use the Symmetry fixture? The answer is: we use this constraint **where object is symmetric about a plane or face**. The live example can be a flat table which can be divided into two or four same sections.

Cyclic Symmetry

This button is used to create symmetry for circular objects. Note that the parts that can be

used for circular symmetry are generally created by revolve tool and are having an axis for symmetry. The steps to use this option are given next.

- Click on the **Cyclic Symmetry** button from the **PropertyManager**. You are asked to select starting face for circular symmetry; refer to [Figure-26](#).

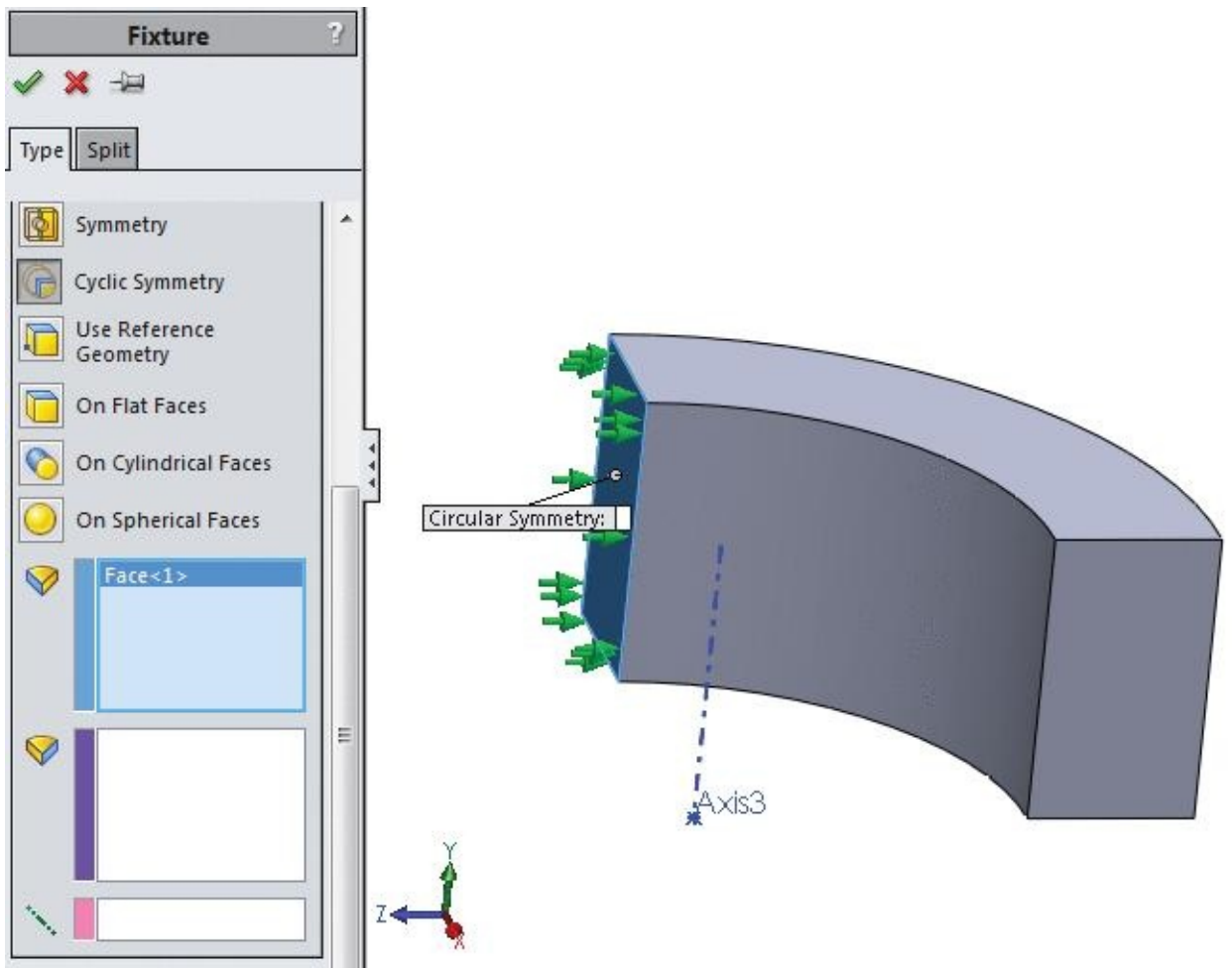


Figure-26. Face selection for cyclic symmetry

- Click in the next box in **PropertyManager**, select the ending face for circular symmetry. Similarly, click in the axis box in **PropertyManager** and then select the axis about which the part is symmetric. Preview of symmetry will be displayed as shown in [Figure-27](#).

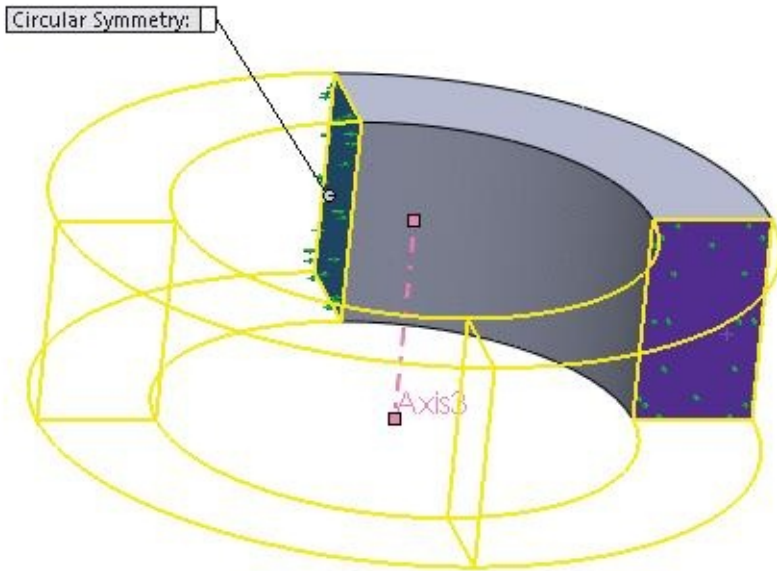


Figure-27. Preview of symmetry fixture

- Click on the **OK** button to apply the circular symmetry fixture.



Where should we use the Cyclic Symmetry fixture? The answer is: we use this constraint **where object can be created by symmetric blocks sharing the same axis.** A live example can be alloy wheels.

Use Reference Geometry

This button is used fix the linear degree of freedom of a body. To apply this fixture, follow the steps given next.

- Click on the **Use Reference Geometry** button from the **Fixture PropertyManager**. The options in the **PropertyManager** will display as shown in [Figure-28](#).

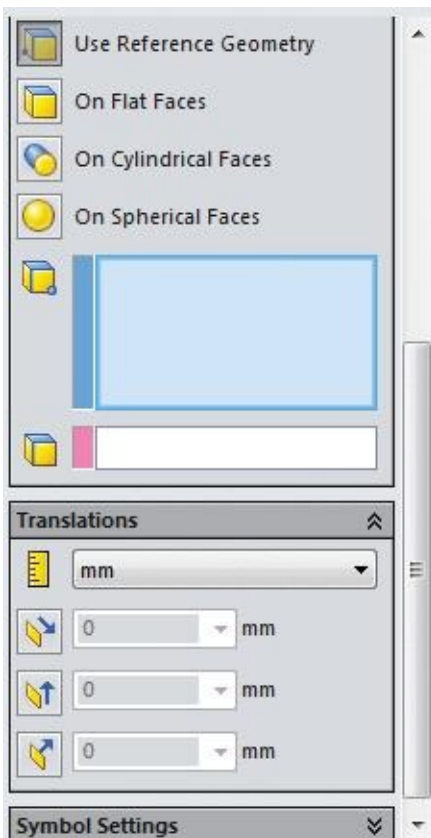


Figure-28. Fixture PropertyManager with Use Reference Geometry button selected

- Select a face on which you want to apply the fixture. Note that you can also select an edge or vertex for fixture.
- Click in the next selection box in **PropertyManager** and select an edge or face to define the direction reference.
- Click in the **Unit** drop-down in the **Translations** drop-down and select the desired unit type.
- Click on the desired direction button from rollout and specify the distance limit.
- The directions for which the buttons are not selected are free.

On Flat Faces

This button is used to fix a flat face in the desired direction. You can also specify the distance limit in specific directions by using this option. The steps to use this button are given next.

- Click on the **On Flat Faces** button from the **Fixture PropertyManager**. The **PropertyManager** will display as shown in [Figure-29](#).

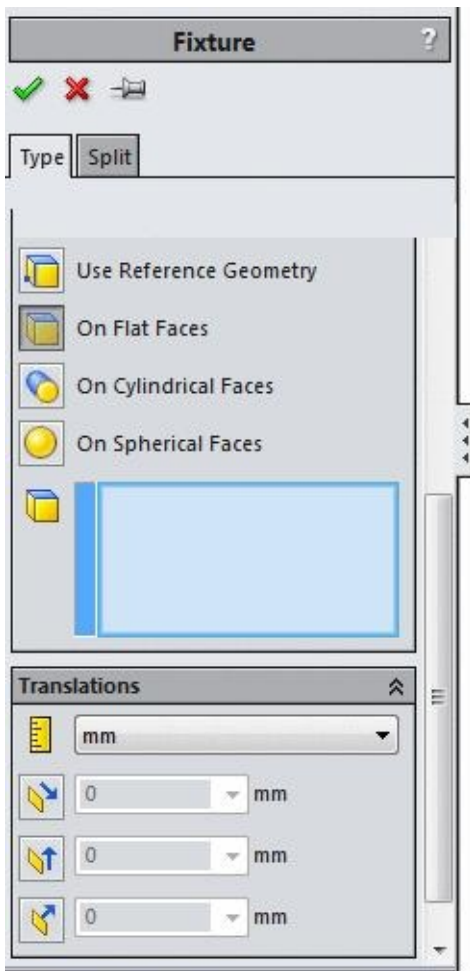


Figure-29. Fixture PropertyManager with On Flat Faces button

- Select the face from the model that you want to fix.
- Click on the button of direction in which you want to specify the distance limit.
- Specify the desired distance in the edit box. If you want to fix the body in the direction then specify the value as **0**.
- You can click on more than one button to constrain more directions; refer to [Figure-30](#).

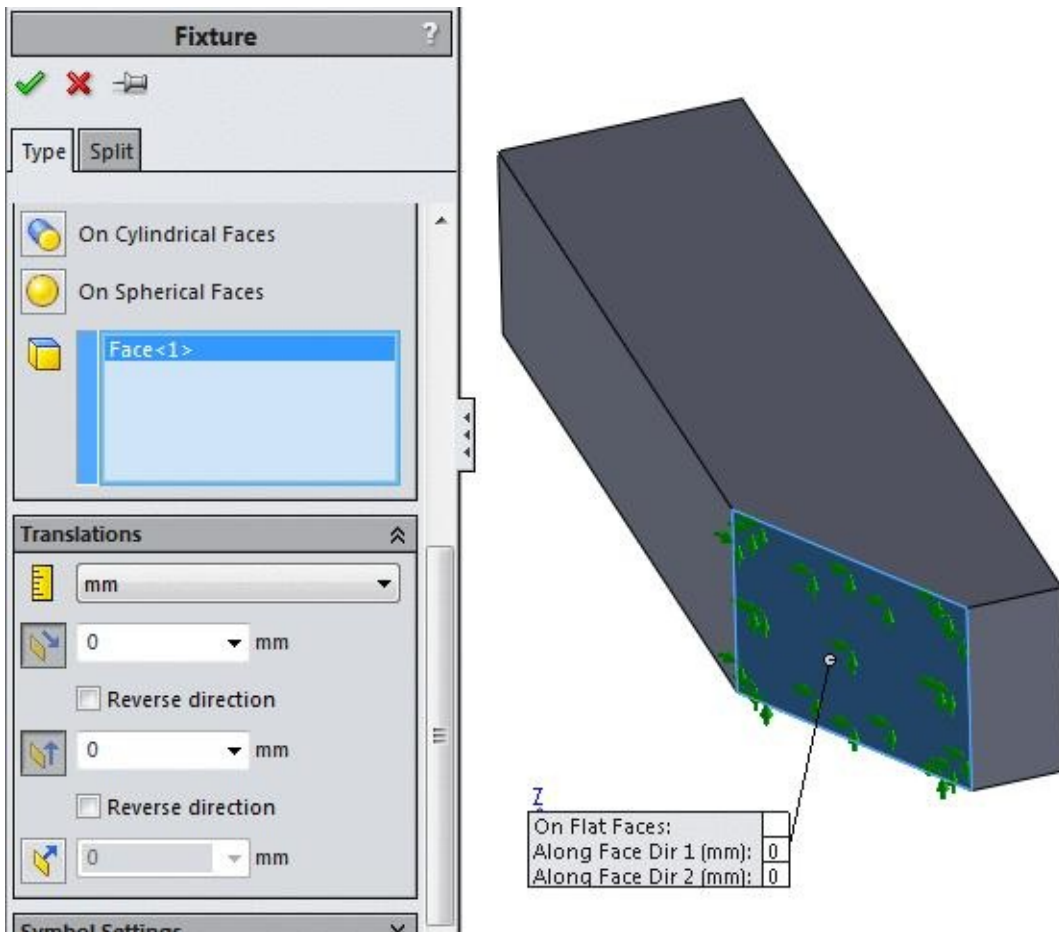


Figure-30. Preview of on flat faces fixture

- Click on the **OK** button from the **PropertyManager** to apply the fixture.

On Cylindrical Faces

This button is used to fix a cylindrical face in the desired direction. This option works in the same way as **On Flat Faces** button works. The steps to use this option are given next.

- Click on the **On Cylindrical Faces** button. The options in the **PropertyManager** will display as shown in [Figure-31](#).



Figure-31. Fixture PropertyManager on selecting On Cylindrical Faces button

- Select the cylindrical face of the model and select the buttons for the directions for which you want to constrain the body movement.
- Specify the distance/rotation limit for the body for the selected direction buttons. Refer to [Figure-32](#).

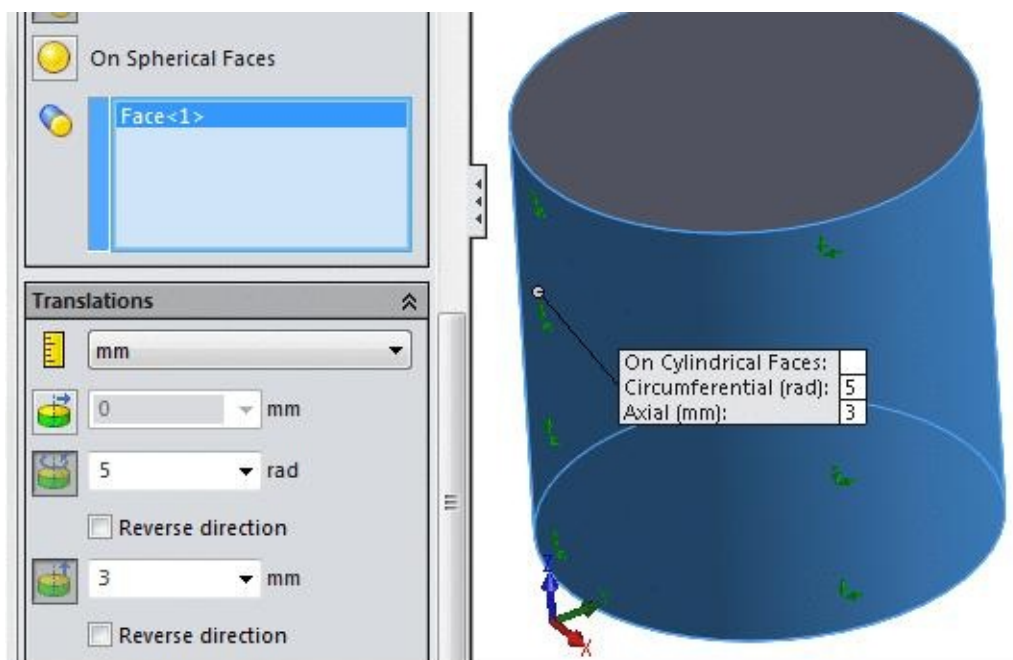


Figure-32. Preview of on cylindrical faces fixture

On Spherical Faces

This button is used to fix a spherical face in the desired directions. This option works in the same way as **On Cylindrical Faces** option. The only difference is that you need to select spherical face for **On Spherical Faces** option. Rest of the procedure is same. Refer to [Figure-33](#).

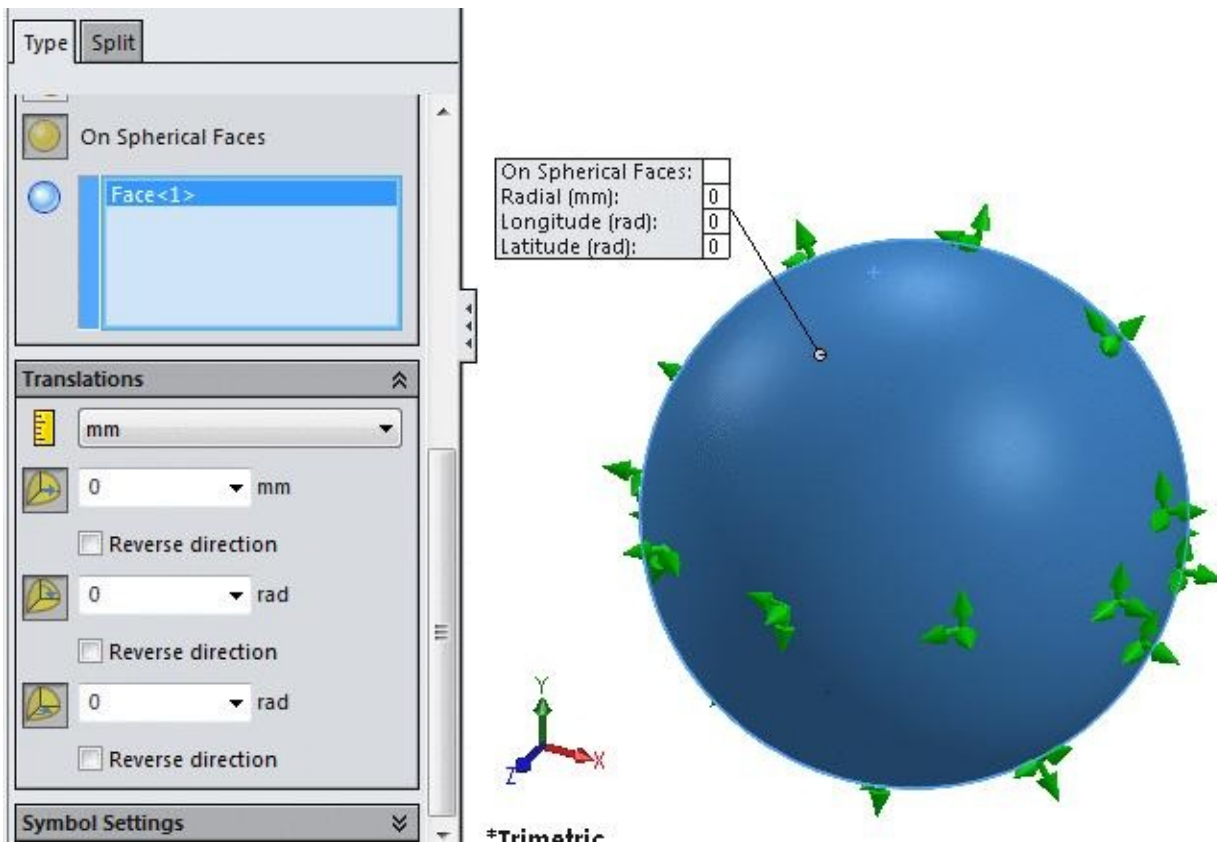


Figure-33. Preview of on spherical faces fixture

Now, you know all the type of fixtures available in SolidWorks Simulation. The next step is to apply the forces on the objects.

EXTERNAL LOAD ADVISOR

This options in this drop-down are used to apply various types of forces on the body. The options in the drop-down are discussed next.

External Load Advisor

This option is used to display the **Simulation Advisor** with the page related to force. The steps to use this option are given next.

- Click on the **External Loads Advisor** option from the **External Load Advisor** drop-

down in the **Simulation** tab of the **Ribbon**. The **Simulation Advisor** will display in the right of the screen as shown in [Figure-34](#).

- Click on the **Add a load** link button from the advisor to apply the load. The options related to forces will displayed in the **Force/Torque PropertyManager**; refer to [Figure-35](#). You can display the same **PropertyManager** by using the **Force** tool from the **External Loads Advisor** drop-down in the **Ribbon**. The method is discussed next.

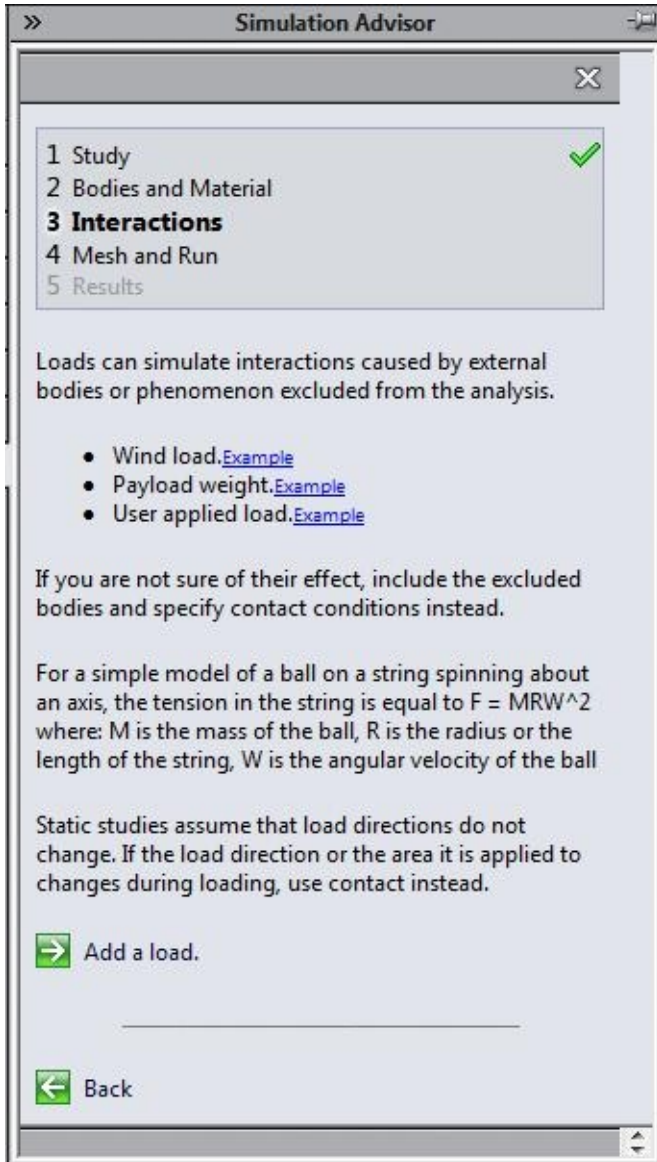


Figure-34. Simulation Advisor with options for load

Force

This is the mostly used option in SolidWorks Simulation for applying forces. To apply the forces, follow the steps given next.

- Click on the **Force** option from the **External Loads Advisor** drop-down in the **Ribbon**. The **Force/Torque PropertyManager** will display as shown in [Figure-35](#).

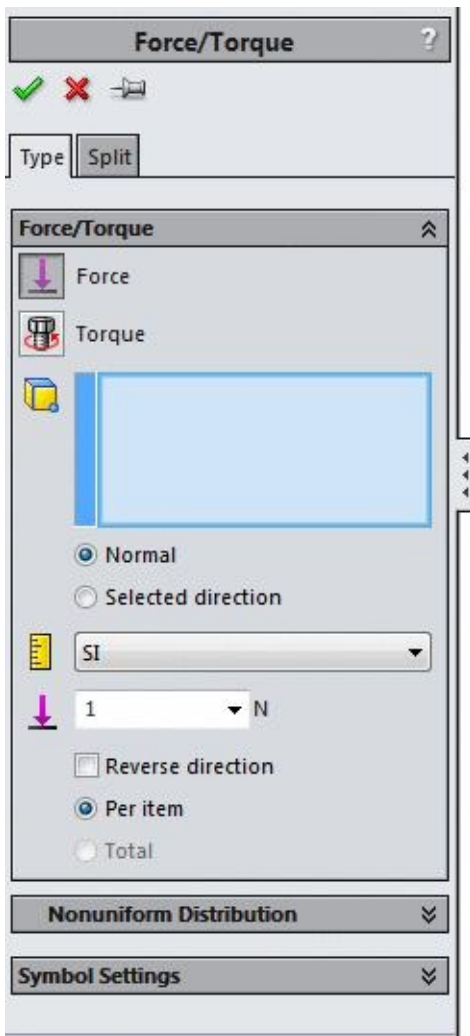


Figure-35. Force Torque PropertyManager

- Select the face perpendicular to which you want to apply the force.
- To change the direction of force, select the **Selected direction** radio button from the **PropertyManager**. You will be asked to select reference for direction via a pink selection box; refer to [Figure-36](#).
- Click in this selection box and select a reference to specify the direction of force; refer to [Figure-37](#). Note that you need to select the force vector buttons from the **Force** rollout in the **PropertyManager** to apply forces.

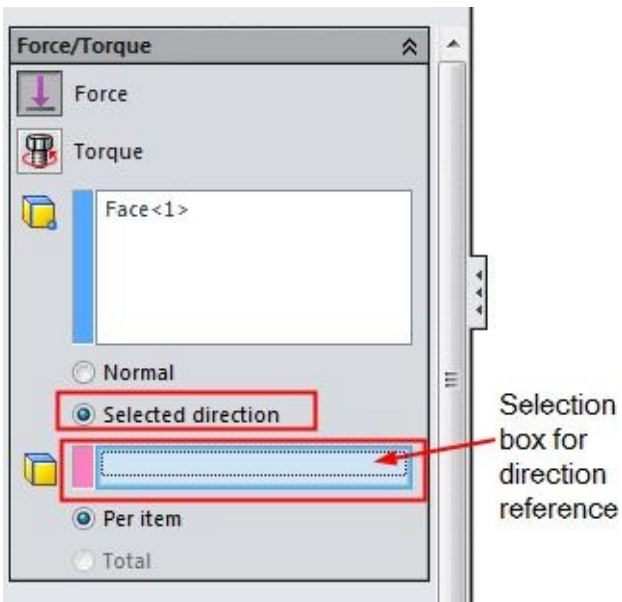


Figure-36. Force in selected direction

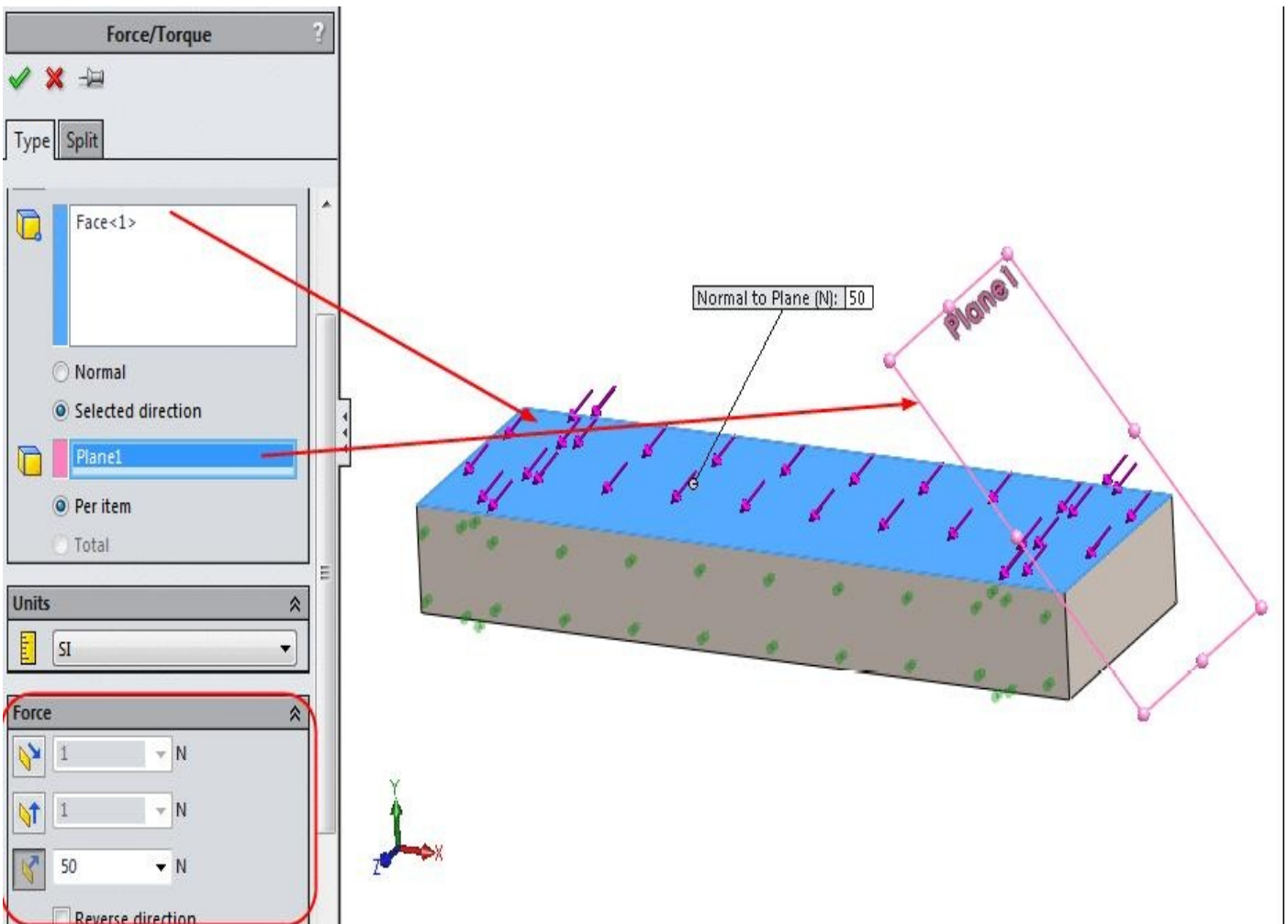


Figure-37. Force applied in selected direction

- If you have select **Normal** radio button in the **PropertyManager** then click in the **Force Value** edit box and specify the desired value.
- You can change the unit type by selecting desired option from the drop-down in the

Force/Torque rollout.

- If you have non uniform distribution of force and have an equation of distribution then click on the check box of **Nonuniform Distribution** rollout. The options related to Non-uniform distribution of force will display; refer to [Figure-38](#).

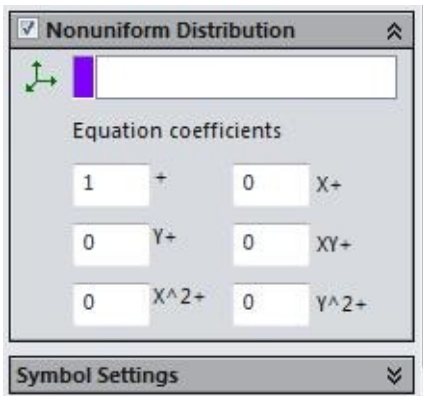


Figure-38. Non uniform force distribution

- Click on the coordinate system that you want to select for specifying x,y, and z components of forces. The equation that can be used in this box is define by the following formula:

$$F(X,Y) = A + B*X + C*Y + D*X*Y + E*X^2 + F*Y^2$$

- In this equation, you need to enter the values of A, B, C, D, E, and F in the edit boxes displayed in the **Nonuniform Distribution** rollout.
- Note that the above equation is only for distribution of force if the force do not reverse the direction during its application.

Torque

Technically Torque is product of force and distance. If force is applied on an object in such a way that it tend to rotate the object then it is called torque. The steps to apply torque are given next.

- Click on the **Torque** button from the **External Loads Advisor** drop-down. The **Force/Torque PropertyManager** will be displayed with the **Torque** button selected by default. Refer to [Figure-39](#).
- Select the round face on which you want to apply torque. Preview of the torque will display; refer to [Figure-40](#).

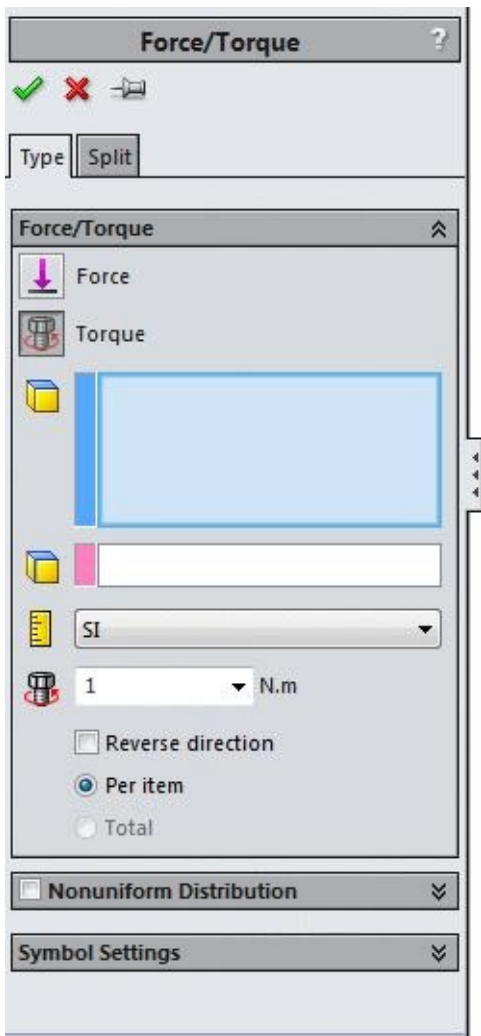


Figure-39. Force Torque PropertyManager with torque options

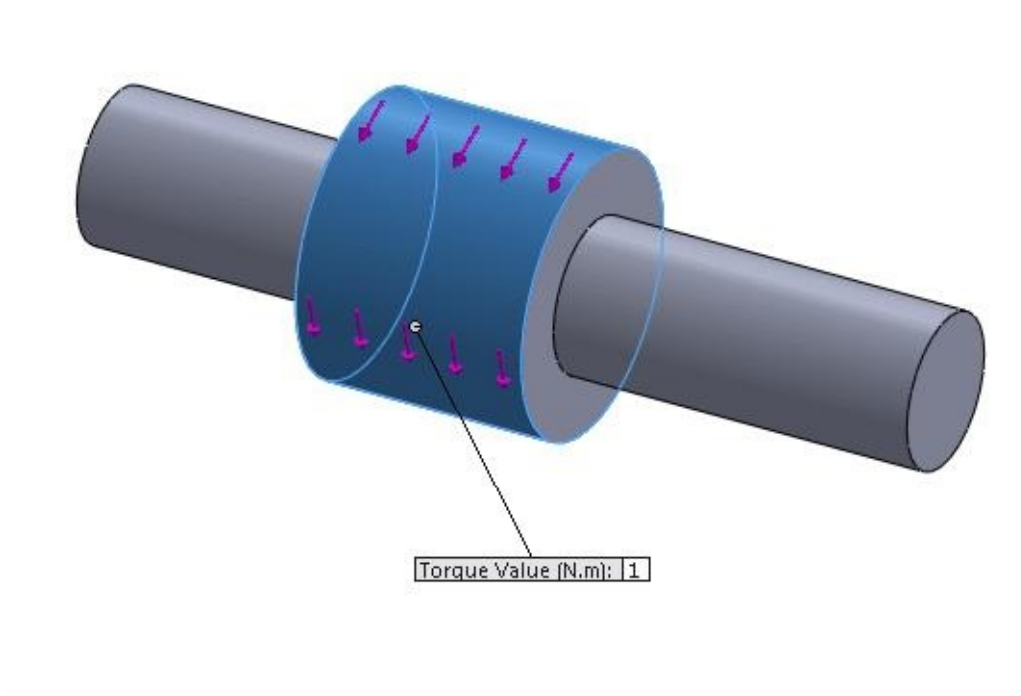


Figure-40. Preview of torque applied

- Specify the desired value of torque in the **Torque Value** edit box of **Force/Torque** rollout.
- The other options in this **PropertyManager** are same as for Force. Click on the **OK** button from the **PropertyManager** to apply torque.

Pressure

Pressure is the force applied per unit area. Generally, we apply pressure when we are analyzing pressure vessels or pipes. The steps to apply pressure are given next.

- Click on the **Pressure** option from the **External Loads Advisor** drop-down. The **Pressure PropertyManager** will display as shown in [Figure-41](#).

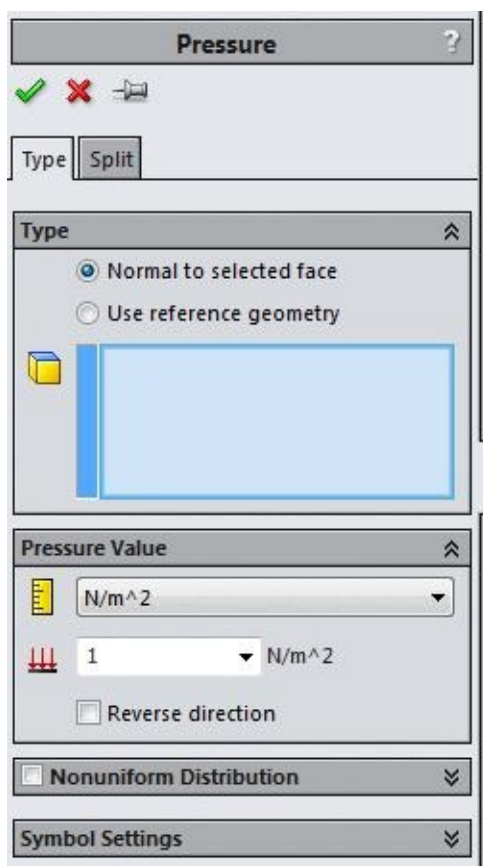


Figure-41. Pressure PropertyManager

- Select the desired radio button from the **Type** rollout to specify the direction of pressure. If you select the **Use reference geometry** radio button then one more selection box will be displayed in the **Type** rollout allowing you to select the direction reference.
- Select the face on which you want to apply the pressure and enter the desired value of pressure in the edit box displayed in the **Pressure Value** rollout.
- If you have selected the **Use reference geometry** radio button then select the direction reference to specify the direction of pressure. Preview of the pressure applied will be displayed; refer to [Figure-42](#).

- Click on the **OK** button from the **PropertyManager** to apply the pressure.

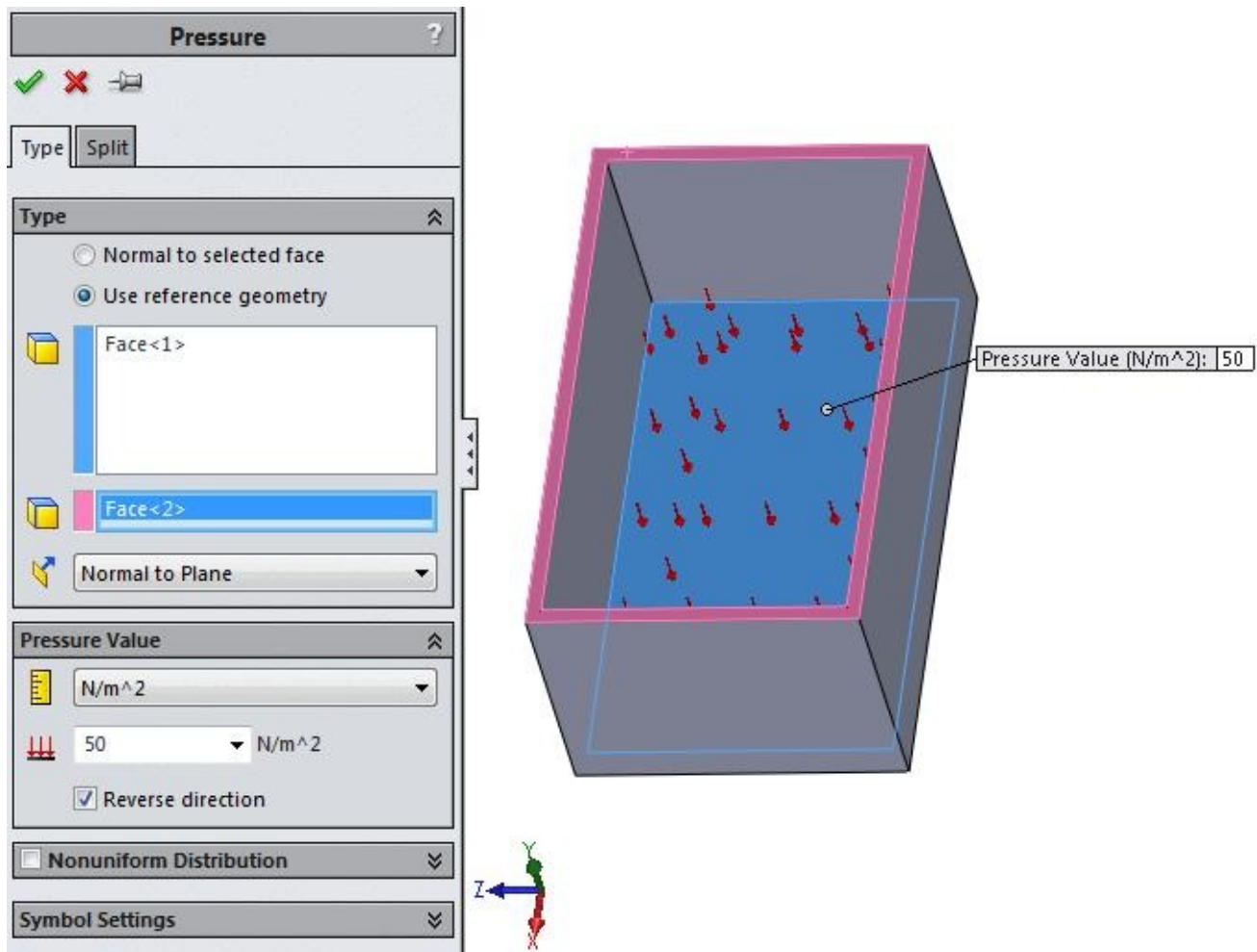


Figure-42. Preview of pressure applied

Gravity

Gravity is the force exerted by earth on each object in earth zone. This is the force that causes objects to fall on the floor. The **Gravity** tool in SolidWorks Simulation is used to apply the gravity force on objects. The steps to use this tool are given next.

- Click on the **Gravity** tool from the **External Loads Advisor** drop-down. The **Gravity PropertyManager** will be displayed as shown in [Figure-43](#).

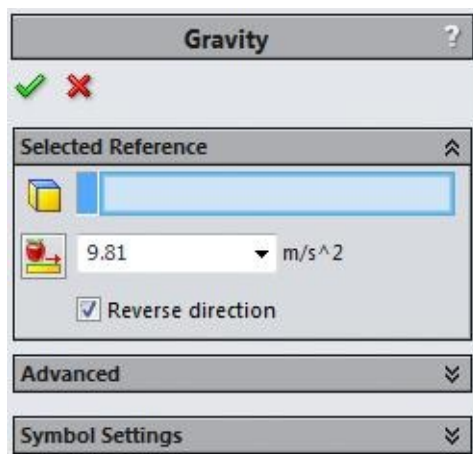


Figure-43. Gravity PropertyManager

- Select the plane/face/edge to which define the direction of gravitational force. Preview of the gravitational force will be displayed; refer to [Figure-44](#).
- Select/clear the **Reverse direction** as per your requirement.

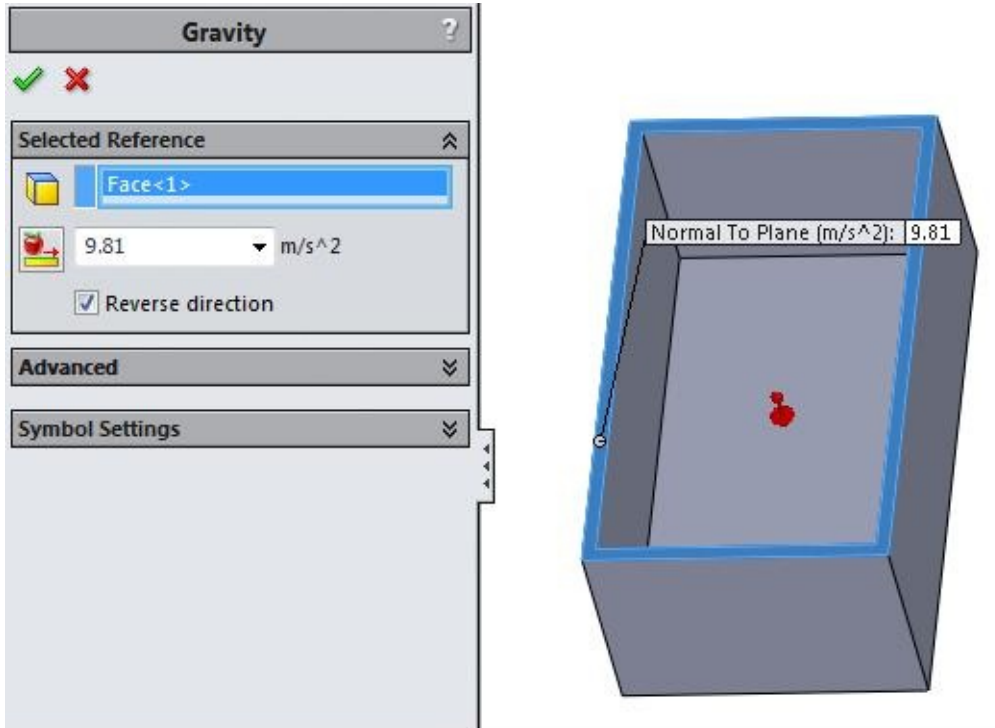


Figure-44. Preview of gravitational force

- If you want to specify more components of gravitational force then expand the **Advanced** rollout and enter the values for two other direction vectors of gravitational force; refer to [Figure-45](#).
- Click **OK** to apply the force.

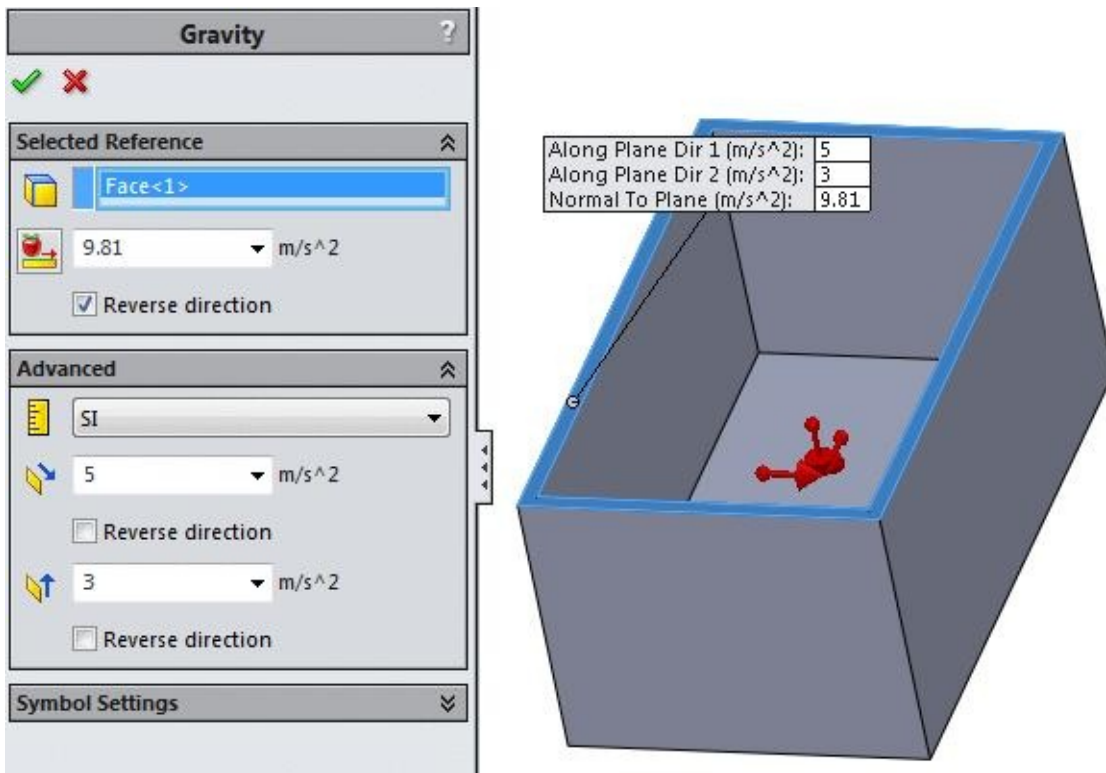


Figure-45. Preview of gravitational force advanced

Centrifugal Force

Centrifugal force is a force which causes the rotating objects to move in opposite direction to the rotation axis, see a Mary-go-round. If you run a Mary-go-round beyond its design limits then you can see linked seats coming outward. This force is generally generated by rotation of bodies. The steps to apply this force are given next.

- Click on the **Centrifugal Force** button from the **External Loads Advisor** drop-down. The **Centrifugal PropertyManager** will display as shown in [Figure-46](#).

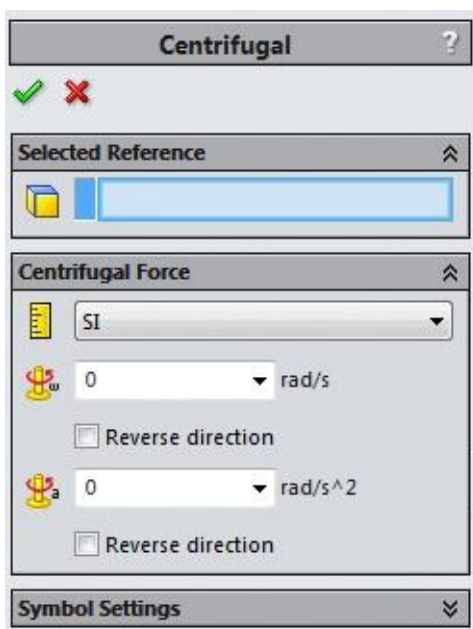


Figure-46. Centrifugal PropertyManager

- Select the round face on which you want to apply the centrifugal force.
- Click in the **Angular Velocity** edit box and specify the angular velocity i.e. omega for the object.
- Click in the **Angular Acceleration** edit box and specify the value of angular acceleration. Preview of the centrifugal force will display; refer to [Figure-47](#).

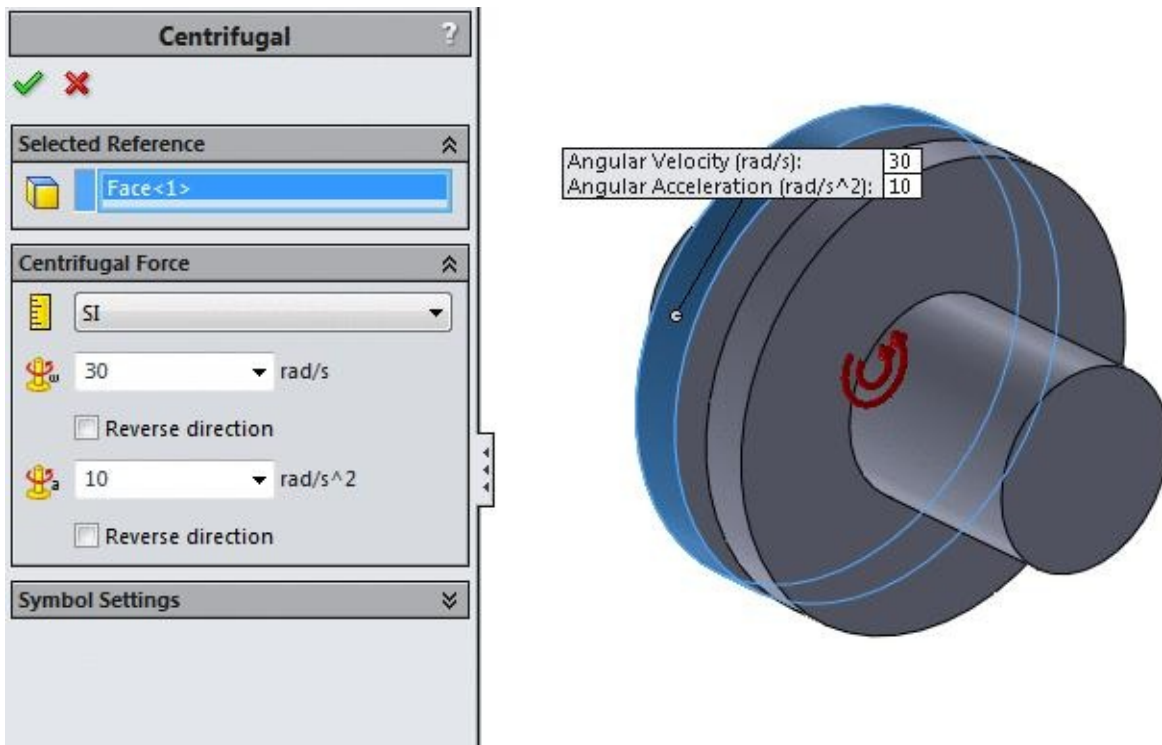


Figure-47. Preview of centrifugal force

- Click on the **OK** button from the **PropertyManager** to apply the force.

Bearing Load

Bearing load is the load exerted by shaft floating in the inner ring of bearing. The bearing load is applied on the round face of bearing and shaft. The steps to apply the load are given next.

- Click on the **Bearing Load** option from the **External Loads Advisor** drop-down. The **Bearing Load PropertyManager** will be displayed as shown in [Figure-48](#).

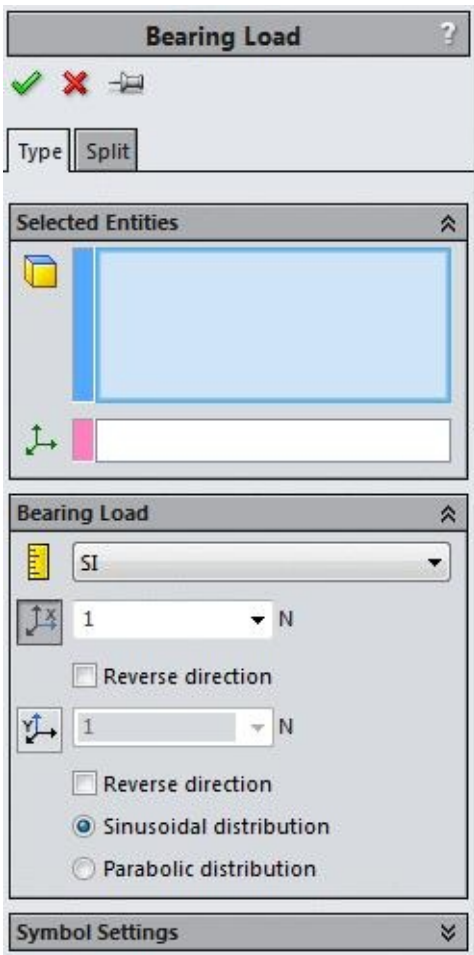


Figure-48. Bearing Load PropertyManager

- Select the faces on which you want to apply the bearing load. Note that you need to select two faces, one from shaft and one from bearing. Also, the selected faces must be half of the full face; refer to [Figure-49](#). Note that you will need the **Split Line** tool to divide the face of bearing and shaft.

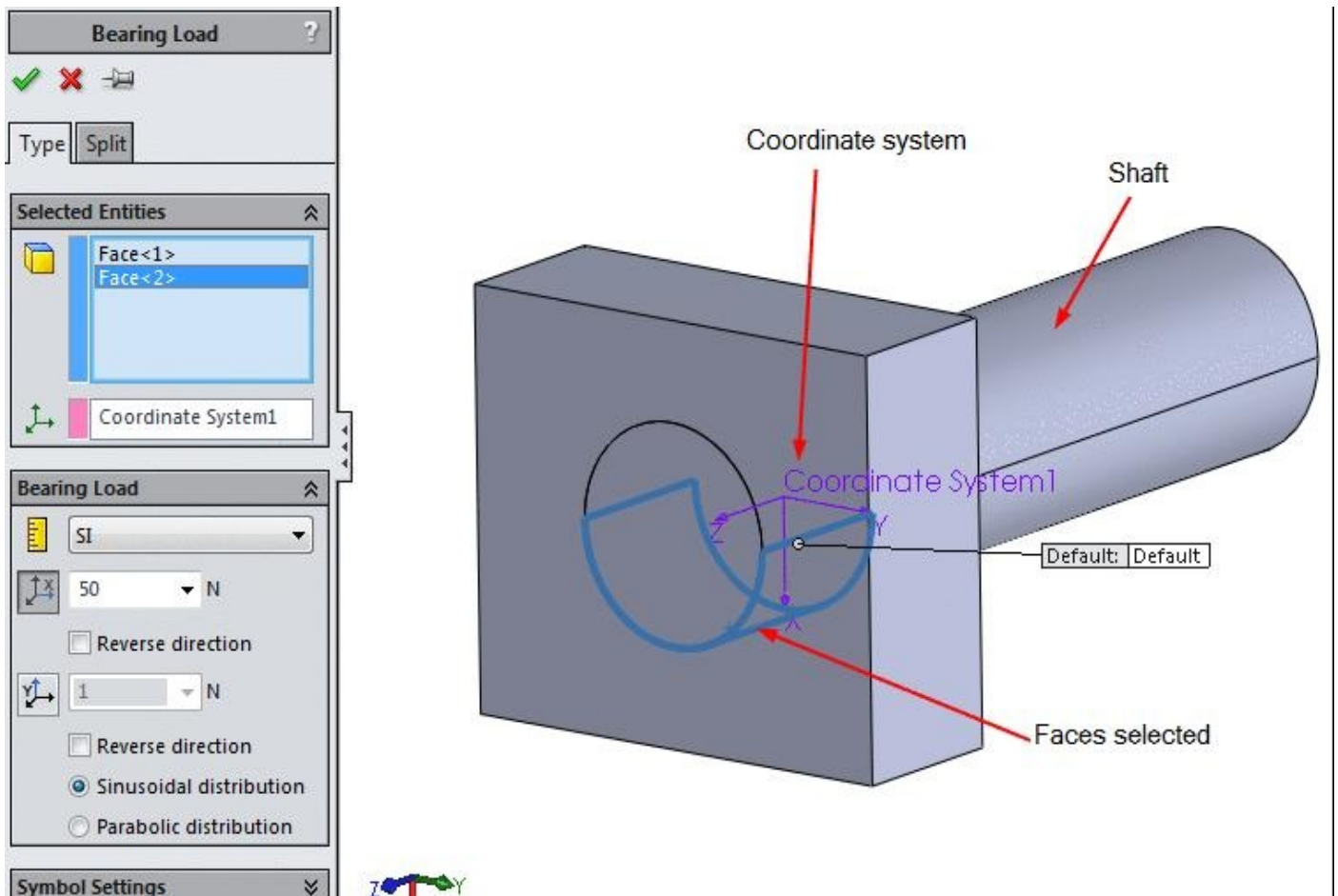



Figure-49. Example for bearing load

- Specify the value of load in X-direction by using the respective edit box in **Bearing Load** rollout.
- If you want to apply load in Y-direction in place of X-direction then click on the  button and specify the value.
- If you want to change the direction as per your requirement then click in the pink selection box and select the desired coordinate system.
- Select the **Sinusoidal distribution** or **Parabolic distribution** radio button as per your requirement.
- Click on the **OK** button from the **PropertyManager** to apply the bearing load.

Remote Loads/Mass

Remote load/mass is used to apply force on an object in such a way that the origin of force is somewhere else but it is also affecting on the selected face; refer to [Figure-50](#).

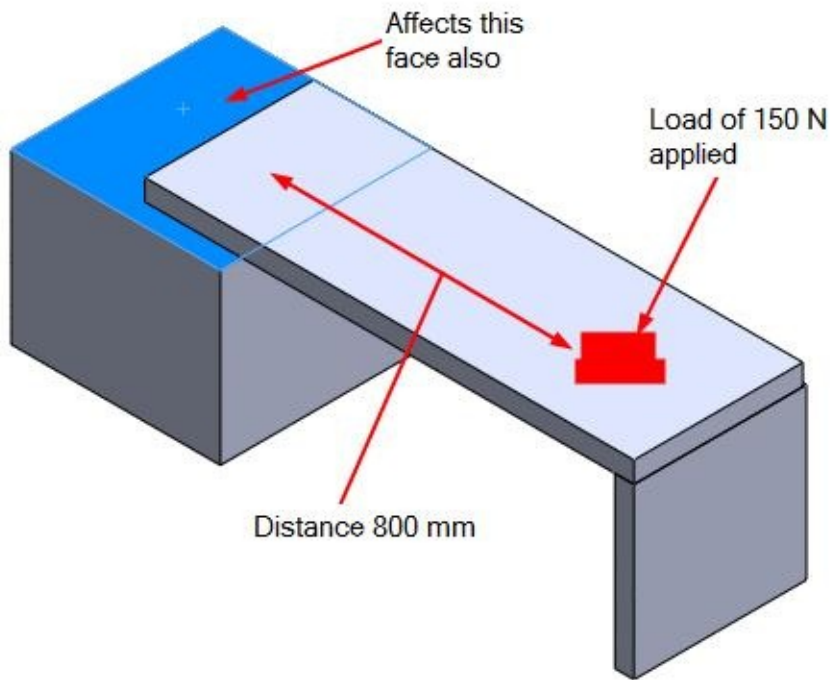


Figure-50. Remote load example

If we apply this load in Solidworks Simulation then it will display as shown in [Figure-51](#).

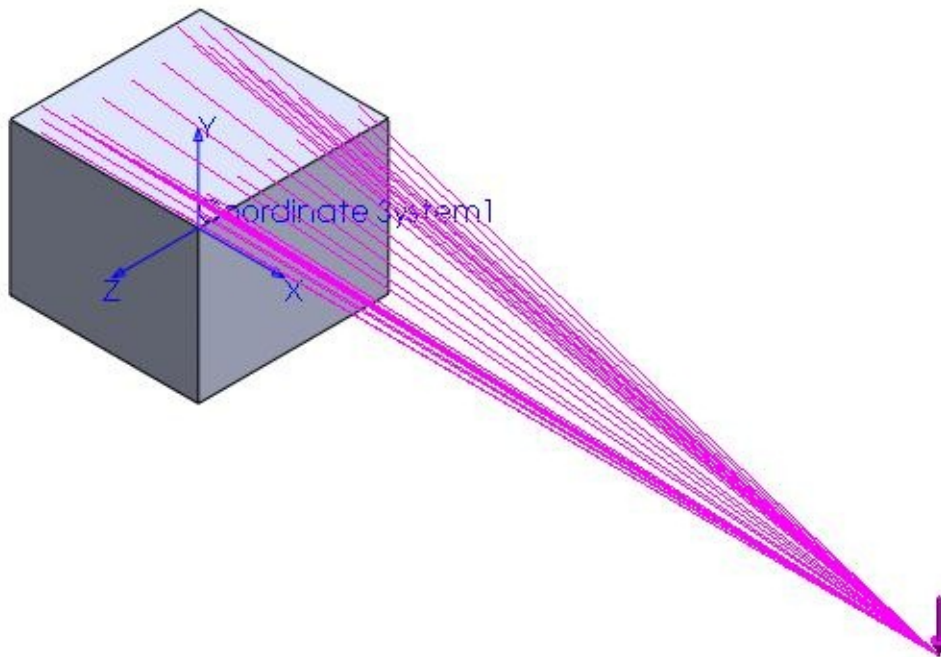


Figure-51. Remote load applied

The procedure to apply remote load is given next.

- Click on the **Remote Load/Mass** option from the **External Loads Advisor** drop-down. The **Remote Loads/Mass PropertyManager** will be displayed as shown in [Figure-52](#).

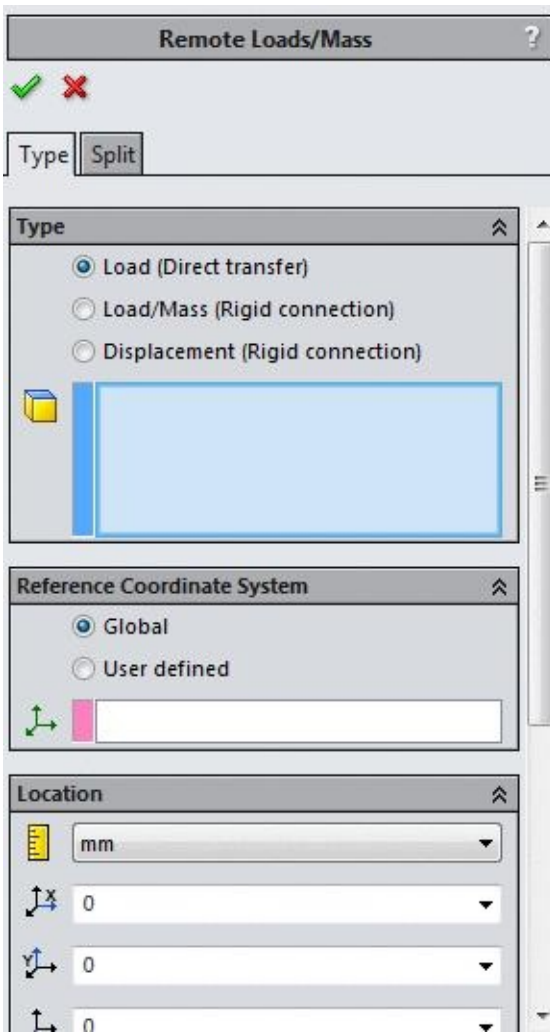


Figure-52. Remote Loads or Mass PropertyManager

- Select the desired radio button from the **Type** rollout to specify the type of remote load/mass you are going to apply. Note that in case of remote load, the exact force working on the selected face is dependent on the total force applied on the action point and distance of action point from the face. If you select the **Load (Direct transfer)** radio button then the exact amount load which comes after calculation is applied on the selected faces. If you select the **Load/Mass(Rigid Connection)** radio button then the load applied on the selected face acts like the load is passed with the help of rigid bars to the selected face. If you select the **Displacement (Rigid Connection)** radio button then it connects the remote location, at which translation and rotations are applied, to the center of the selected faces by a rigid bar. The selected entities deform accordingly.
- After selecting the desired radio button from **Type** rollout, select a reference coordinate system after selecting the **User Defined** radio button from the **Reference Coordinate System** rollout.
- Specify the location of remote point by using the edit boxes in the **Location** rollout in the **PropertyManager**; refer to [Figure-53](#).

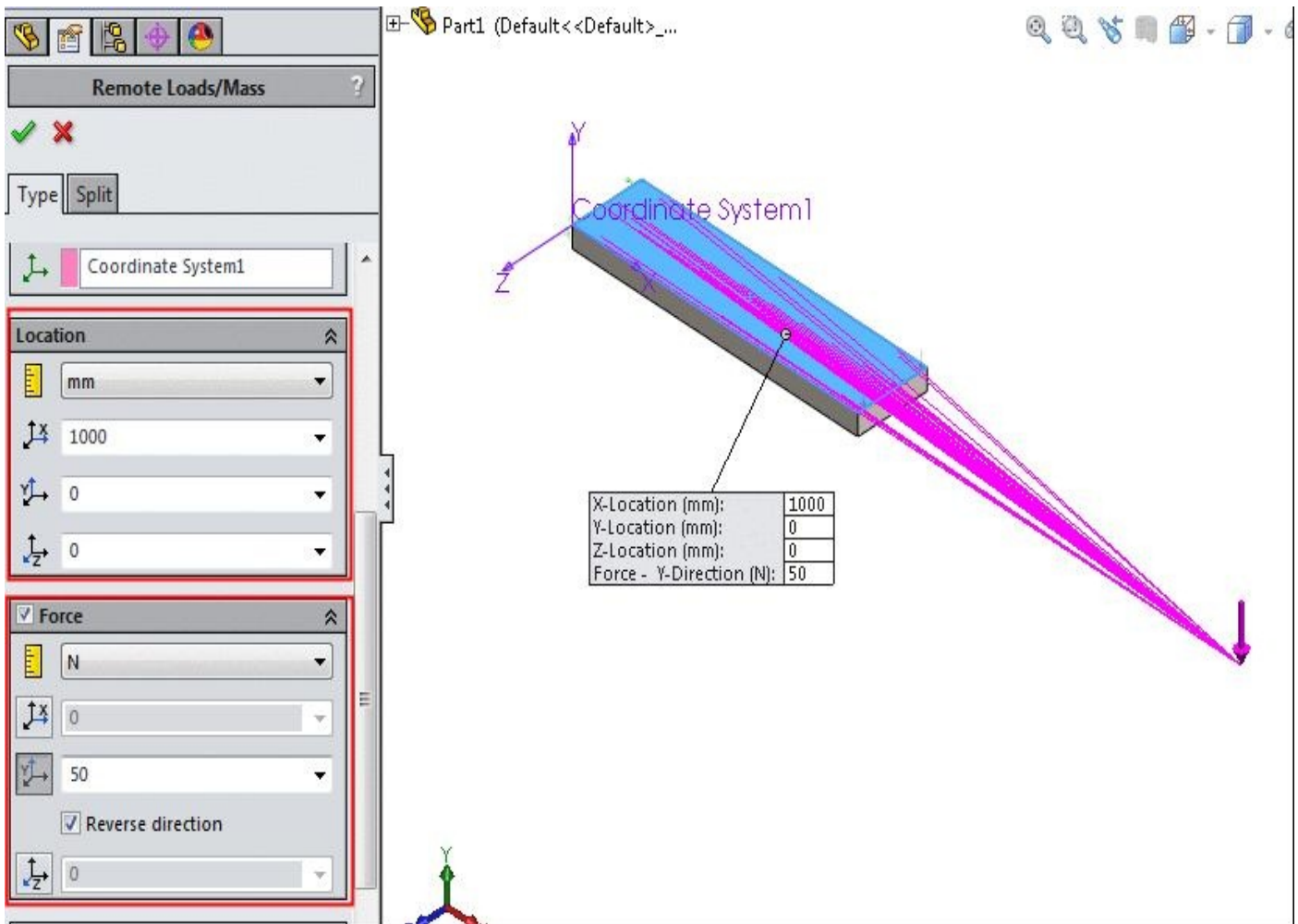


Figure-53. Remote load location

- Similarly, specify the value of force in each direction and then click on the **OK** button from the **PropertyManager** to apply the forces. You can also specify the moment of force by using the options in the **Moment** rollout.

Distributed Mass

Using this option, you can add the mass of the objects that are not created in the models but their mass makes effect on the analysis. The steps to use this option are given next.

- Click on the **Distributed Mass** button from the **External Loads Advisor** drop-down. The **Distributed Mass PropertyManager** will display; refer to [Figure-54](#).

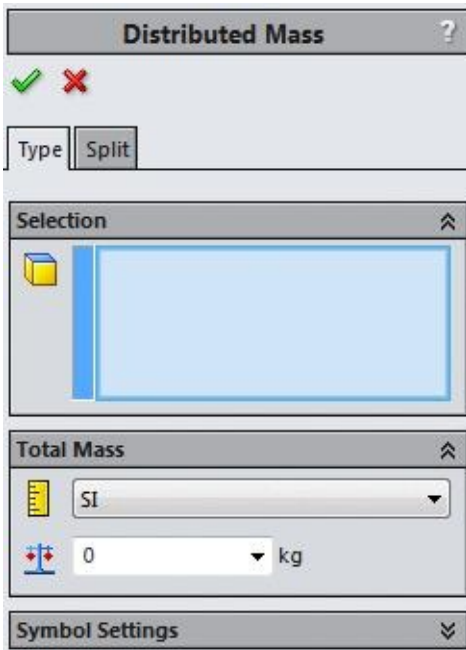


Figure-54. Distributed Mass PropertyManager

- Select the face on which you want to add mass of remote object.
- Click in the **Distributed Mass** edit box and specify the desired value. Preview of the applied mass will be displayed; refer to [Figure-55](#).

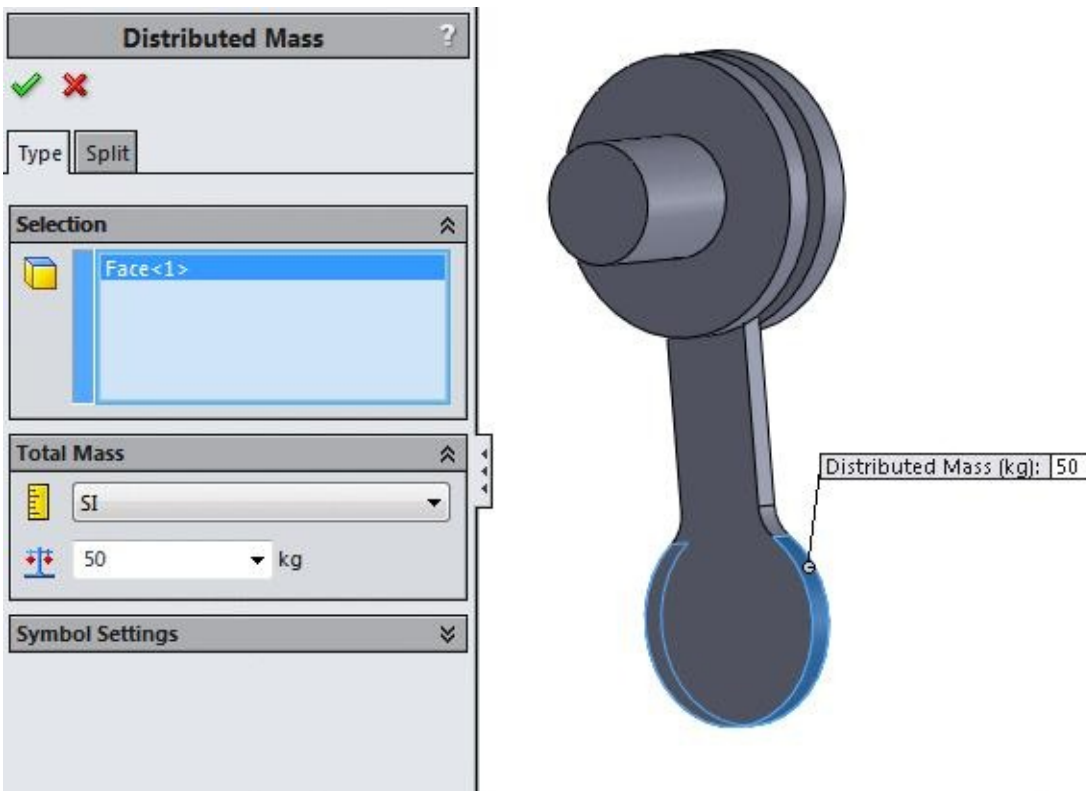


Figure-55. Preview of distributed mass

- Click on the **OK** button from the **PropertyManager** to apply the distributed mass.

Temperature

This option is used to set temperature of the selected face. This option is mostly used in thermal analyses. The steps to use this option are given next.

- Click on the **Temperature** button from the **External Loads Advisor** drop-down. The **Temperature PropertyManager** will display as shown in [Figure-56](#).

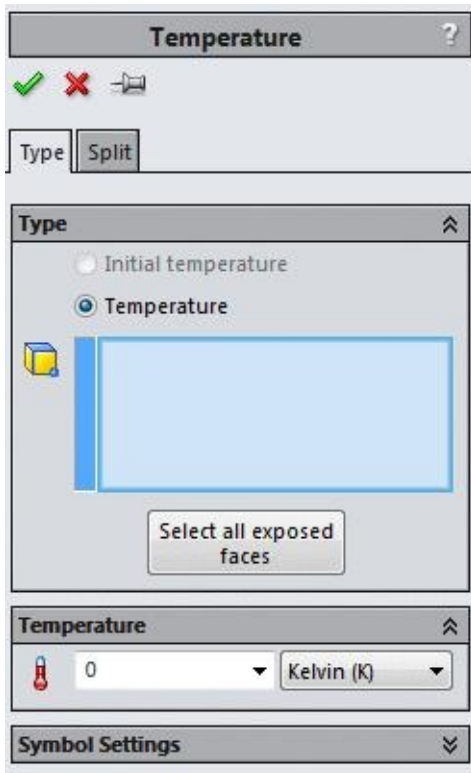


Figure-56. Temperature PropertyManager

- Select the face for which you want to set the temperature.
- Click in the **Temperature** edit box of the **Temperature** rollout and specify the desired value in the edit box. Preview of the applied temperature will be displayed; refer to [Figure-57](#).

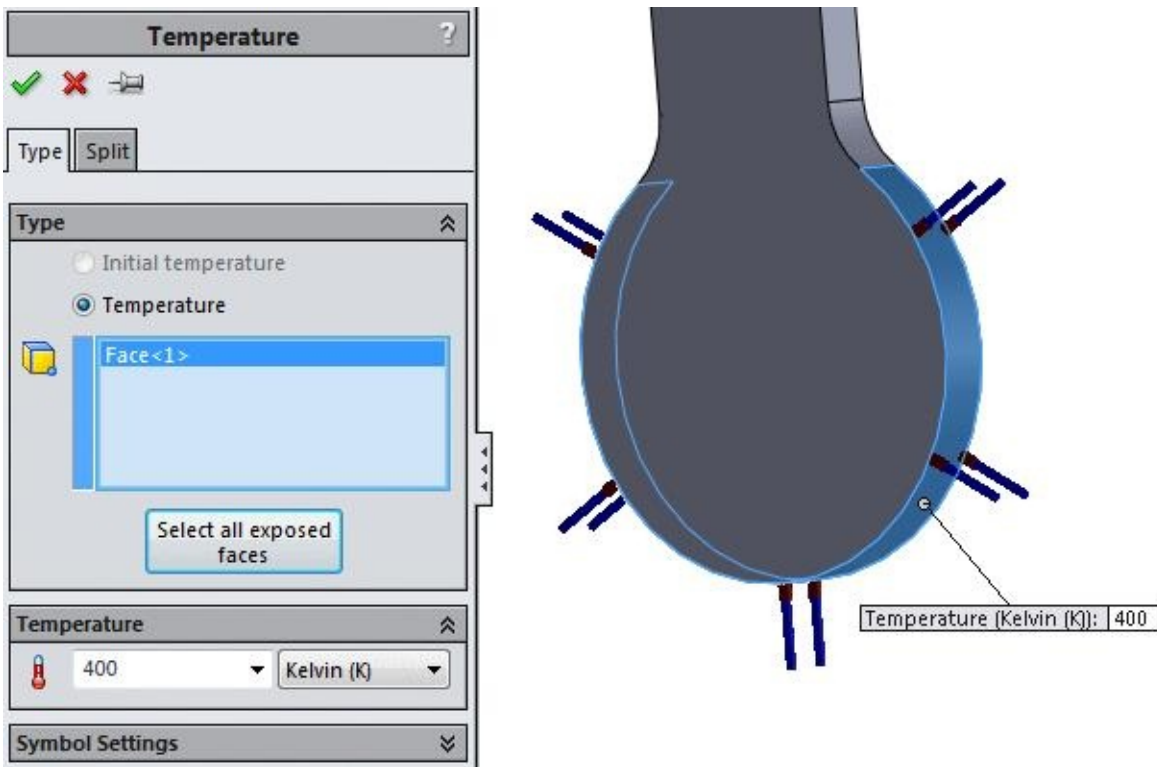


Figure-57. Preview of temperature

- You can apply temperature on all the exposed faces by selecting the **Select all exposed faces** button.

Flow Effects/Thermal Effects

These options are used to add the fluid flow or thermal effects in the current study. The steps to specify the flow effect setting and thermal effect settings are given next.

- Click on the **Flow Effects** or **Thermal Effects** button from the **External Loads Advisor** drop-down. A dialog box will be displayed with the **Flow/Thermal Effects** tab selected; refer to [Figure-58](#).

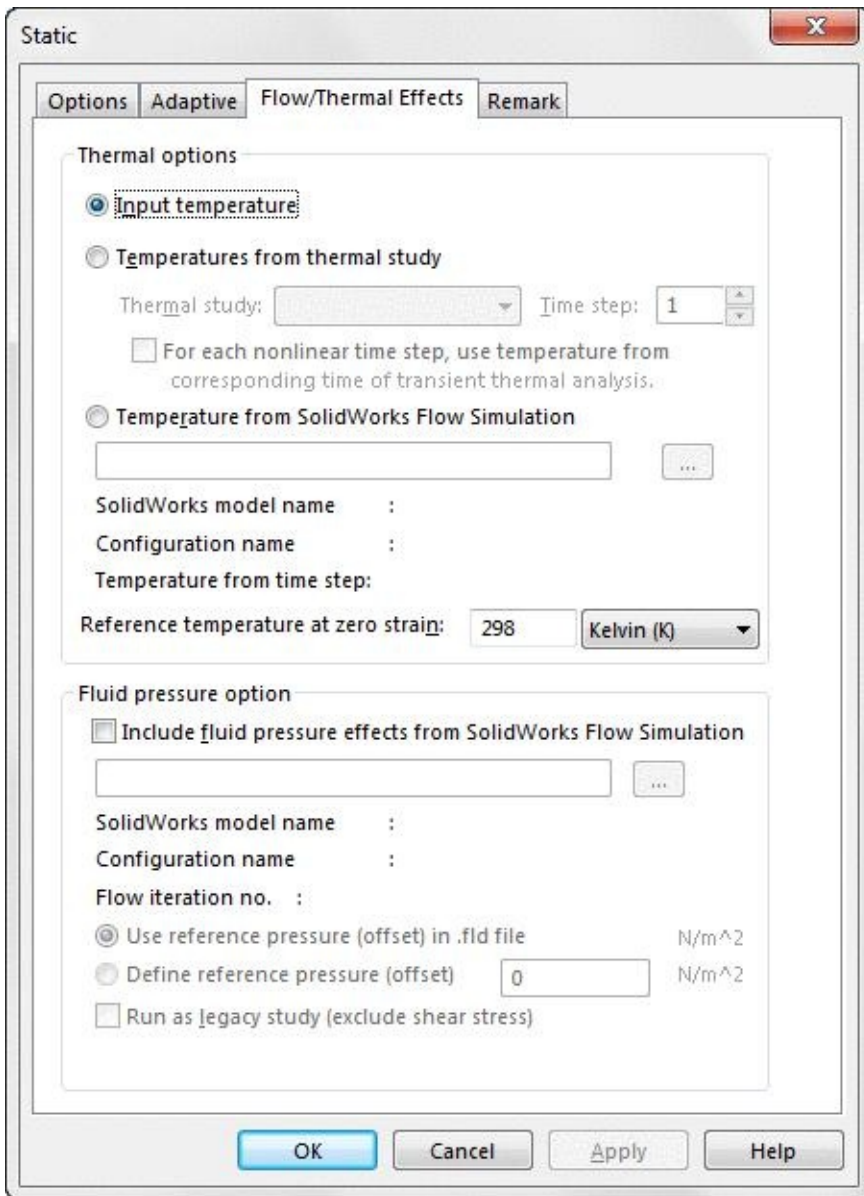


Figure-58. Flow Thermal Effects tab of Static dialog box

- This dialog box is divided into two areas: **Thermal options** and **Fluid pressure options**. The options in these areas are discussed next.

Thermal options

The options in this area are used to specify the source for temperatures in the active study. There are three options to do that:

Input Temperature

Select this radio button to manually specify the temperature for the study. 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

Select this option to use the output temperature of a thermal study already performed. On selecting this radio button, the options below it will become active. Select the desired thermal study from the drop-down to get its output temperature. You can also set the time for getting the temperature at a specific point of time. For a transient thermal study, select either a Time step number (to import a single temperature) or For each nonlinear time step, use temperature from corresponding time of transient thermal analysis.

Temperatures from SolidWorks Flow Simulation

Select this option to use the output temperature of a completed **SolidWorks Flow Simulation** on the same configuration. Click on this radio button and then click on the **Browse** button to select the result file of flow simulation. Select the desired result file (*.fld) that has been generated by Flow Simulation. Model name, configuration name, and time step number associated with the specified file are displayed.

Reference temperature at zero strain

This edit box is used to set the temperature at which no strains exist in the model.

Fluid pressure option

The options in this area are used to import fluid pressure loads from a SolidWorks Flow Simulation results file.

Include fluid pressure effects from Flow Simulation

Select this option to use the output pressure from a completed SolidWorks Flow Simulation results file. On selecting this option the Browse button will become active below it. Click on the button and select the desired result file (*.fld) that has been generated by Flow Simulation. For proper acquisition of pressure loads from Flow Simulation, the active study and the **Flow Simulation** study should be associated with the same configuration. To create the *.fld file, click **Flow Simulation > Tools > Export Results to Simulation**.

Use reference pressure (offset) in .fld file

Select this option to use the reference pressure defined in the Flow Simulation results file.

Define reference pressure (offset)

Select this option to specify a reference pressure directly. This pressure value is subtracted from the imported pressure values.

Run as legacy study (exclude shear stress)

Using this option you can imports only the normal component of the pressure load from the Flow Simulation.

If you are analyzing an assembly, then you need to specify the type of connections between various components of the assembly. The options to specify connections for assembly are available in the **Connections Advisor** drop-down which is discussed next.

CONNECTIONS ADVISOR

This options in this drop-down are used to apply various types of connections between assembly components. You can also specify the connections between components and ground by using these options. These options are discussed next.

Connections Advisor

This option is used to display the page of **Simulation Advisor** that is related to connections between assembly components. The steps to use this option are given next.

- Click on the **Connections Advisor** button from the **Connections Advisor** drop-down in the **Ribbon**. The **Simulation Advisor** will be displayed with the options to apply connections; refer to [Figure-59](#).

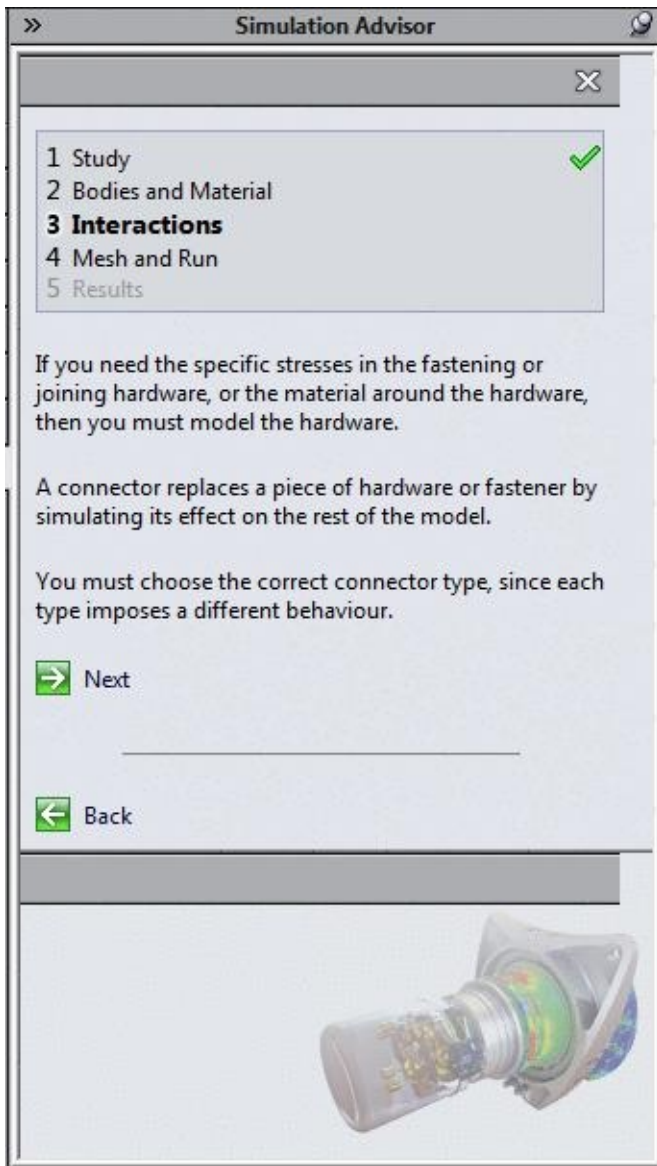


Figure-59. Simulation Advisor with options for connections

- Click on the **Next** button from the **Simulation Advisor**. The list of options that can be used for connections are displayed.
- Select the desired option and follow the instructions given by the software. Note that the options in the **Simulation Advisor** are discussed one by one in the next section.

Contact Sets

This option is used to specify the contact type between two components or a component and ground. The steps to use this option are given next.

- Click on the **Contact Sets** button from the **Connections Advisor** drop-down. The **Contact Sets PropertyManager** will display as shown in [Figure-60](#).

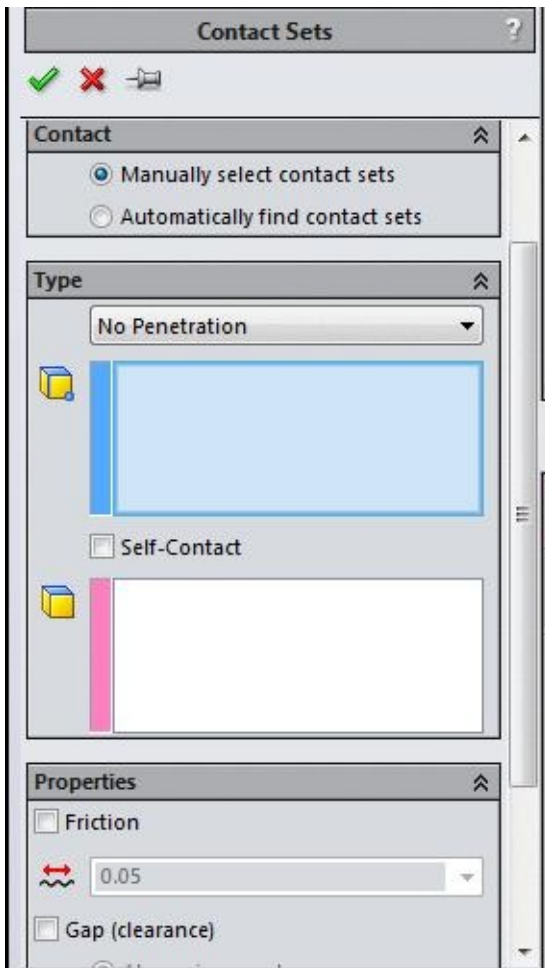


Figure-60. Contact Sets PropertyManager

- Select the **Automatically find contact sets** option from the **Contact** rollout if you want the software to automatically decide the type of contact to be applied.
- Select the **Manually select contact sets** option to specify the connections manually.
- On selecting the **Manually select contact sets** option, two selection boxes will become available.
- Select the desired option from the drop-down in the **Type** rollout. The options in this drop-down and the process of using these options are given next.

No Penetration

This option is used when you want the faces to be in touch by specified gap value but do not want them to penetrate through each other. The steps to use this option are given next.

- Select the **No Penetration** option from the drop-down in the **Type** rollout. The options in the **PropertyManager** will be displayed as shown in [Figure-61](#).

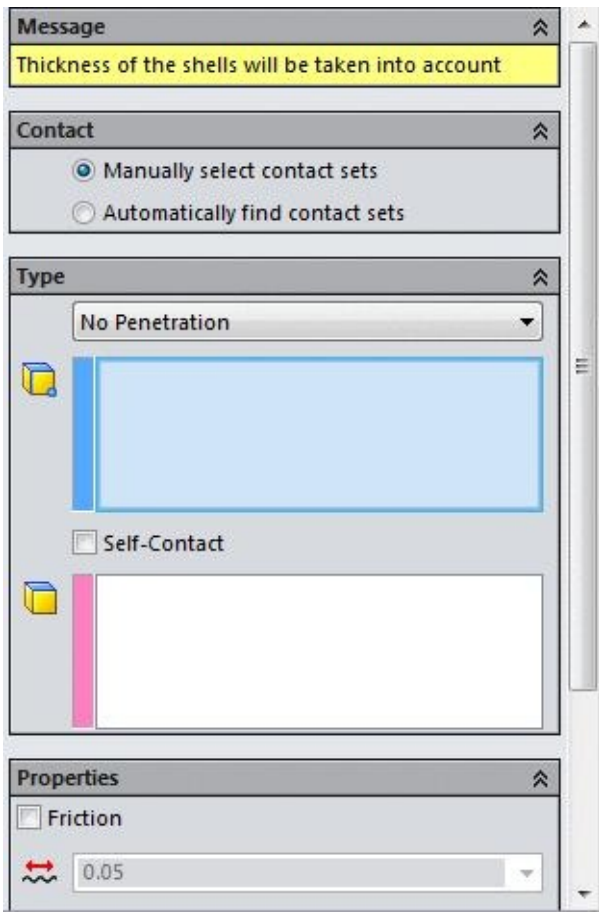


Figure-61. Options in Contact Sets PropertyManager on selecting No Penetration option

- Click in the blue selection box and select the face/edge/point of the first part that you want to specify for connection.
- Click in the pink box and select the face/plane of second component or ground to create the connection with.
- Click on the **Friction** check box and specify the coefficient of friction for contact.
- Click on the **Gap (clearance)** check box and specify the desired gap between the two components or component and ground.
- You can select the **Self-Contact** check box, if you want your object to not intersect with itself under the application of force; refer to [Figure-62](#). This check box is new addition in SolidWorks 2015.

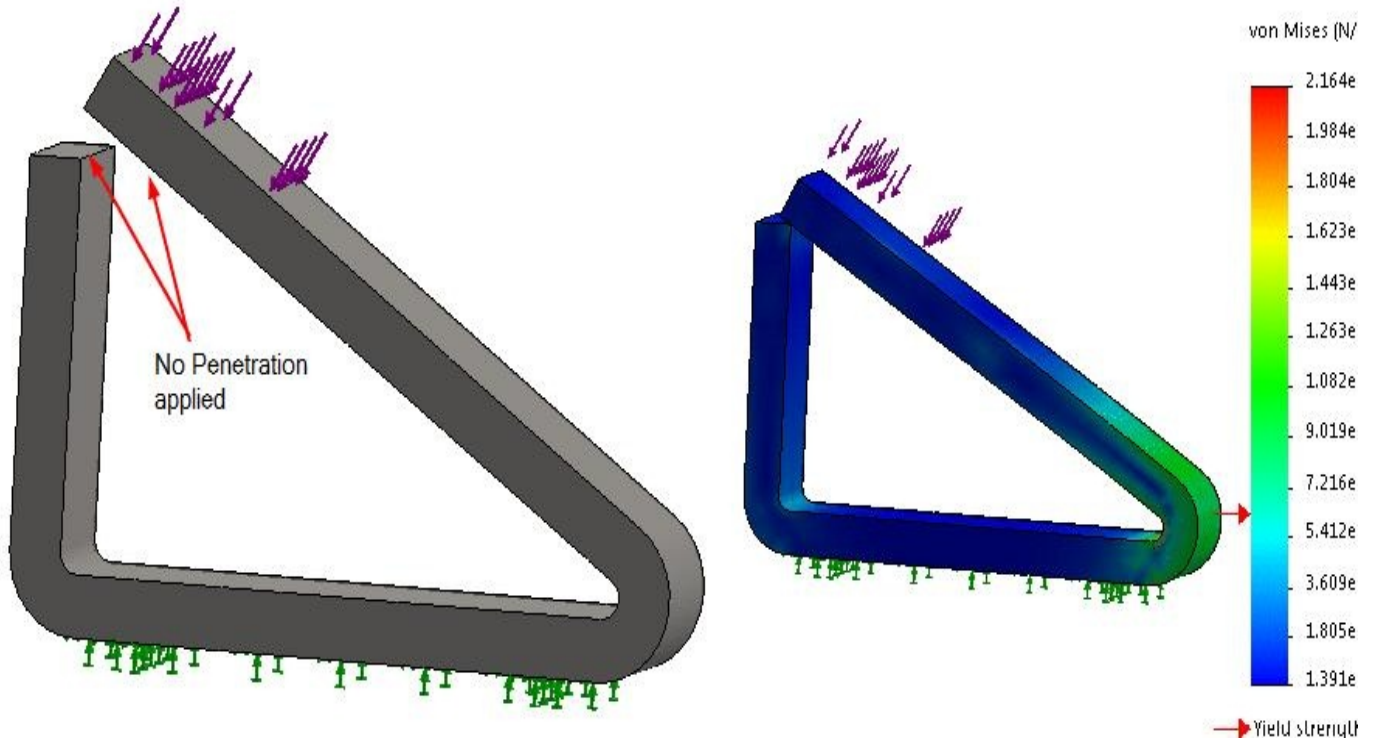


Figure-62. Result of load with Self Contact checkbox

- Click on the check box for **Advanced** rollout and specify the method of contact between the components by selecting the appropriate option.

Bonded

This option is used to attach one component to the other such that both the components are permanently bonded to each other. In other words, effect of force on one component will also be passed on the bonded component perfectly. The steps to use this option are given next.

- Select the **Bonded** option from the drop-down in the **Type** rollout. The options in the **PropertyManager** will be displayed as shown in [Figure-63](#).

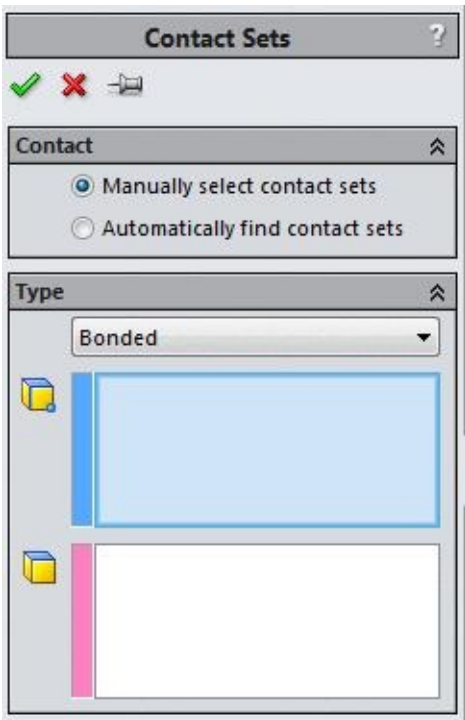


Figure-63. Options in Contact Sets PropertyManager on selecting Bonded option

- Click in the blue box and select the face/edge/vertex of the first component.
- Click in the pink box and select the face/plane of the second component. Preview of the bonded connection will be displayed; refer to [Figure-64](#).

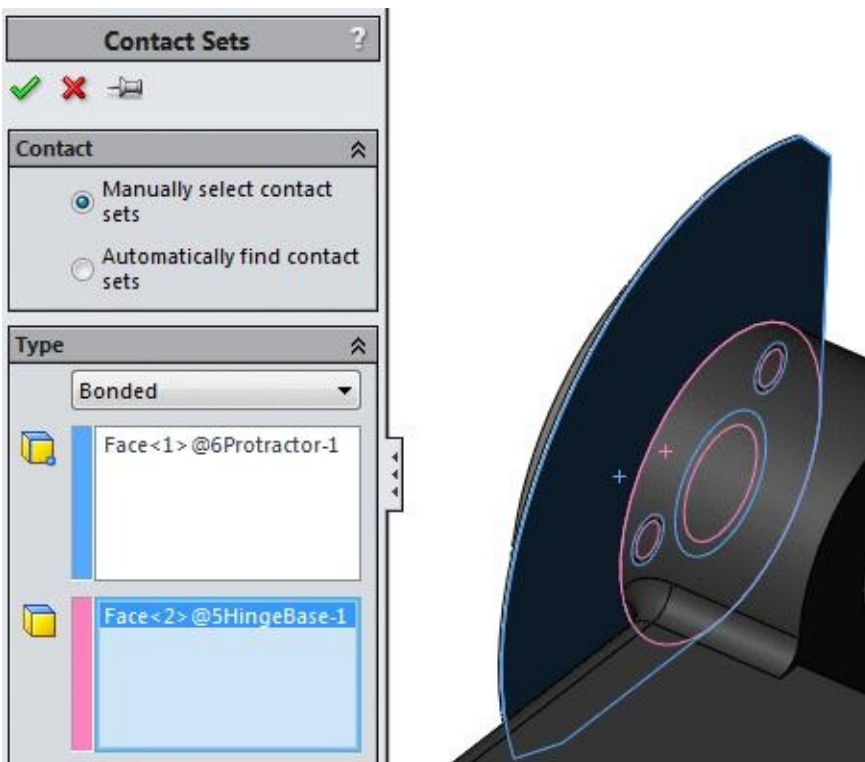


Figure-64. Preview of bonded connection

- Click on the **OK** button from the **PropertyManager** to create the connection.

Allow Penetration

This option is used allow penetration of one object with the other object. Note that this option is generally used when the objects are ductile. The steps to used this option are given next.

- Select the **Allow Penetration** option from the drop-down in the **Type** rollout. The options in the **PropertyManager** will be displayed as shown in [Figure-65](#).

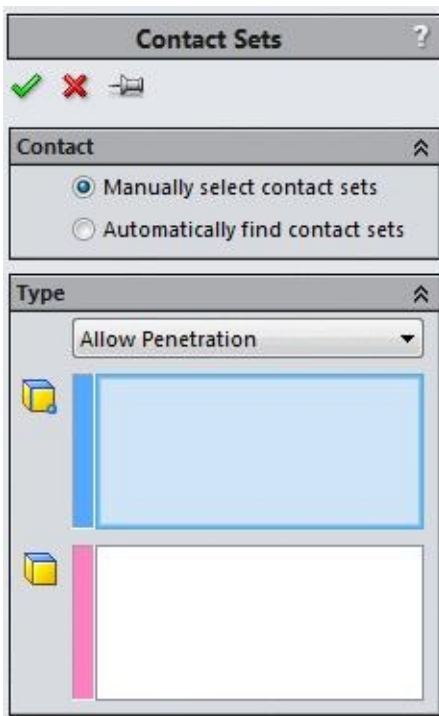


Figure-65. Options in Contact Sets PropertyManager on selecting Allow Penetration option

- Click in the blue selection box and select the face/edge/vertex of the first component.
- Click in the pink selection box and select the face/plane of the second component.
- Click on the **OK** button from the **PropertyManager**.

Shrink Fit

Shrink fitting is a process in which a shaft is inserted in a hole that is heated up. When the hot hole cools down it strongly arrests the shaft. If you need to specify such a connection in the assembly, then this option is used. The steps to use this option are given next.

- Click on the **Shrink Fit** option from the drop-down in the **Type** rollout of **PropertyManager**. The **PropertyManager** will be displayed as shown in [Figure-66](#).

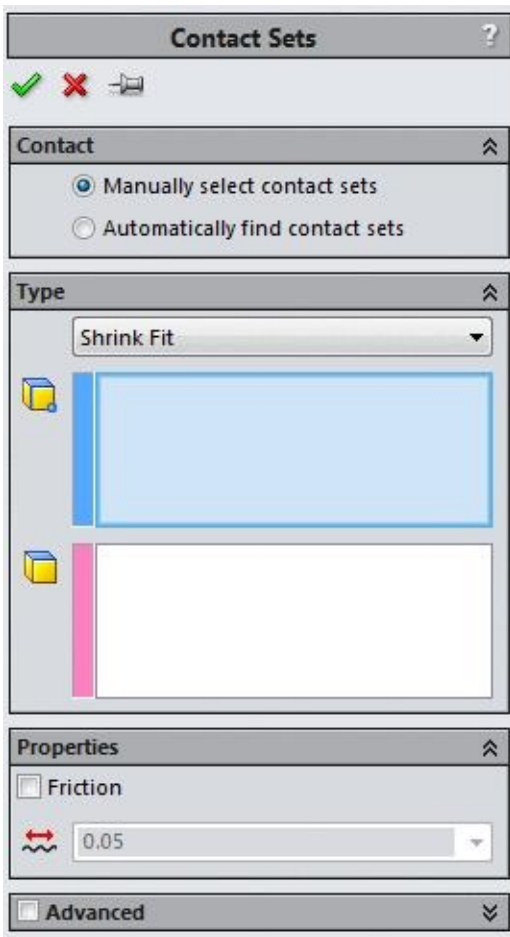


Figure-66. Options in Contact Sets PropertyManager on selecting Shrink Fit option

- You can apply this option in the same way as discussed earlier. You can also specify friction coefficient by selecting the **Friction** check box.

Virtual Wall

Using this option, you can make a component attached to a virtual wall. In this case, the body reacts as if it is in contact with a rigid/flexible wall. The steps to apply this contact are given next.

- Select the **Virtual Wall** option from the drop-down in the **Type** rollout. The options in the **PropertyManager** will display as shown in [Figure-67](#).

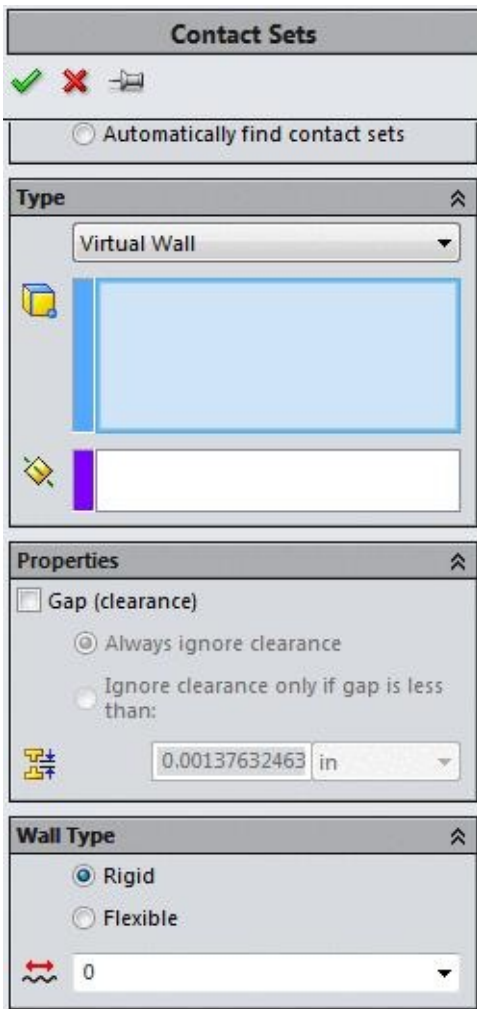


Figure-67. Options in Contact Sets PropertyManager on selecting Virtual Wall option

- Select the face that you want to attach with the virtual wall.
- Click in the violet selection box to collect the plane.
- Select the desired plane that you want to make as virtual wall.
- Click on the **Gap (clearance)** check box to specify the clearance gap.
- Click on the **Rigid** radio button to create a rigid wall and specify the friction coefficient for the wall in the edit box displayed in **Wall Type** rollout.
- Click on the **Flexible** radio button to create a flexible wall.
- Specify the axial stiffness and tangential stiffness in the edit boxes available in the **Wall Stiffness** rollout.

Component Contact

While working on an assembly, this option is generally preferred over the other options available for connections. Using this option you can specify the contact between two or more bodies along their all shared faces. The steps to use this option are given next.

- Click on the **Component Contact** button from the **Connections Advisor** drop-down. The **PropertyManager** will display as shown in [Figure-68](#).

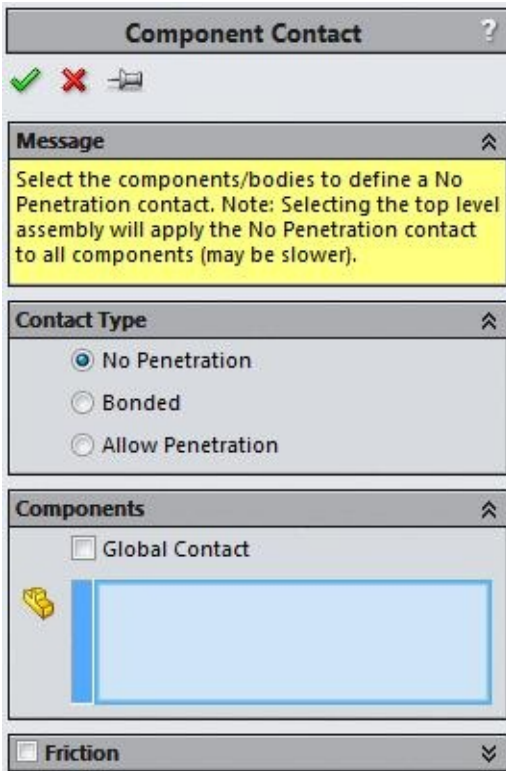


Figure-68. Component Contact PropertyManager

- Select the desired radio button from the **Contact Type** rollout.
- Select the **Global Contact** check box to apply the selected contact type to all the components of the assembly or select the components one by one to specify the contact type.
- Select the check box of **Friction** rollout and specify the friction coefficient.
- If you have selected the **Bonded** radio button from the **Contact Type** rollout then you can specify the type of mesh for the components.
- If you select the **Compatible mesh** radio button then a combined mesh for the all the components bonded will be created. If you select the **Incompatible mesh** radio button then individual mesh will be created for each of the bonded components.
- Click on the **OK** button from the **PropertyManager** to apply the contact.

Contact Visualization Plot

After applying various contacts on components of the assembly, you can review these contacts by selecting **Contact Visualization Plot** button from the **Connections Advisor** drop-down in the **Ribbon**. The steps to use this option are given next.

- Click on the **Contact Visualization Plot** button from the **Connections Advisor** drop-

down. The **Contact Visualization Plot PropertyManager** will display as shown in [Figure-69](#).

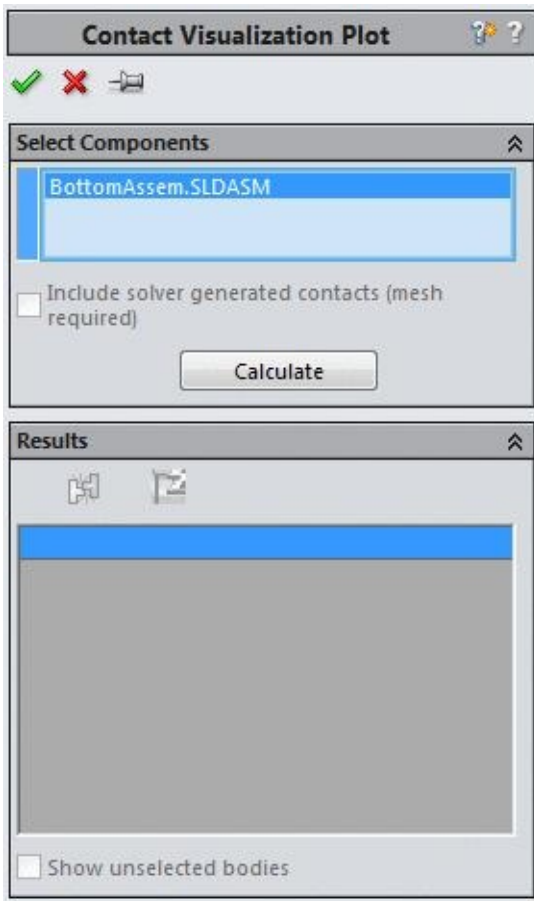


Figure-69. Contact Visualization Plot PropertyManager

- Click on the **Calculate** button from the **PropertyManager** to display the connections applied in the assembly. The list of connections will be displayed in the **Results** rollout.
- Click on the **OK** button from the **PropertyManager** to exit the tool.

Spring

This option is used to connect two components in such a way that they behave like joined by a spring. The steps to connect two components by using the **Spring** button are given next.

- Click on the **Spring** button from the **Connections Advisor** drop-down. The **Spring PropertyManager** will display as shown in [Figure-70](#).

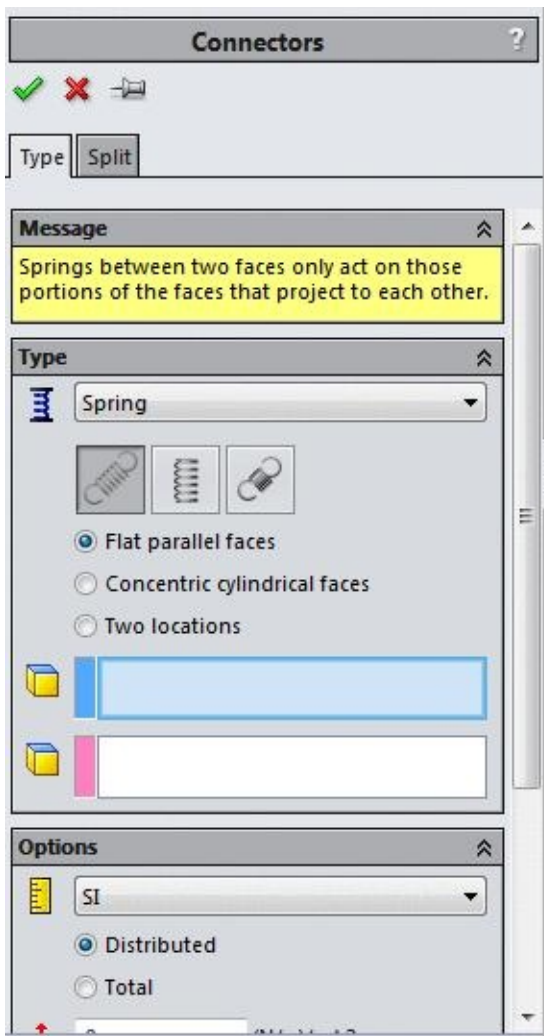


Figure-70. Connectors PropertyManager with Spring option

- Select the desired spring type from the **Type** rollout. There are three type of springs available in the **Type** rollout; **Compression-expansion**, **compression**, and **expansion**.
- Select the desired radio button from **Flat parallel faces**, **Concentric cylindrical faces**, and **Two locations** radio buttons.
- Select the entities as per the selected radio button.
- Specify the parameters related to spring in the **Options** rollout. Preview of the spring will be displayed; refer to [Figure-71](#).

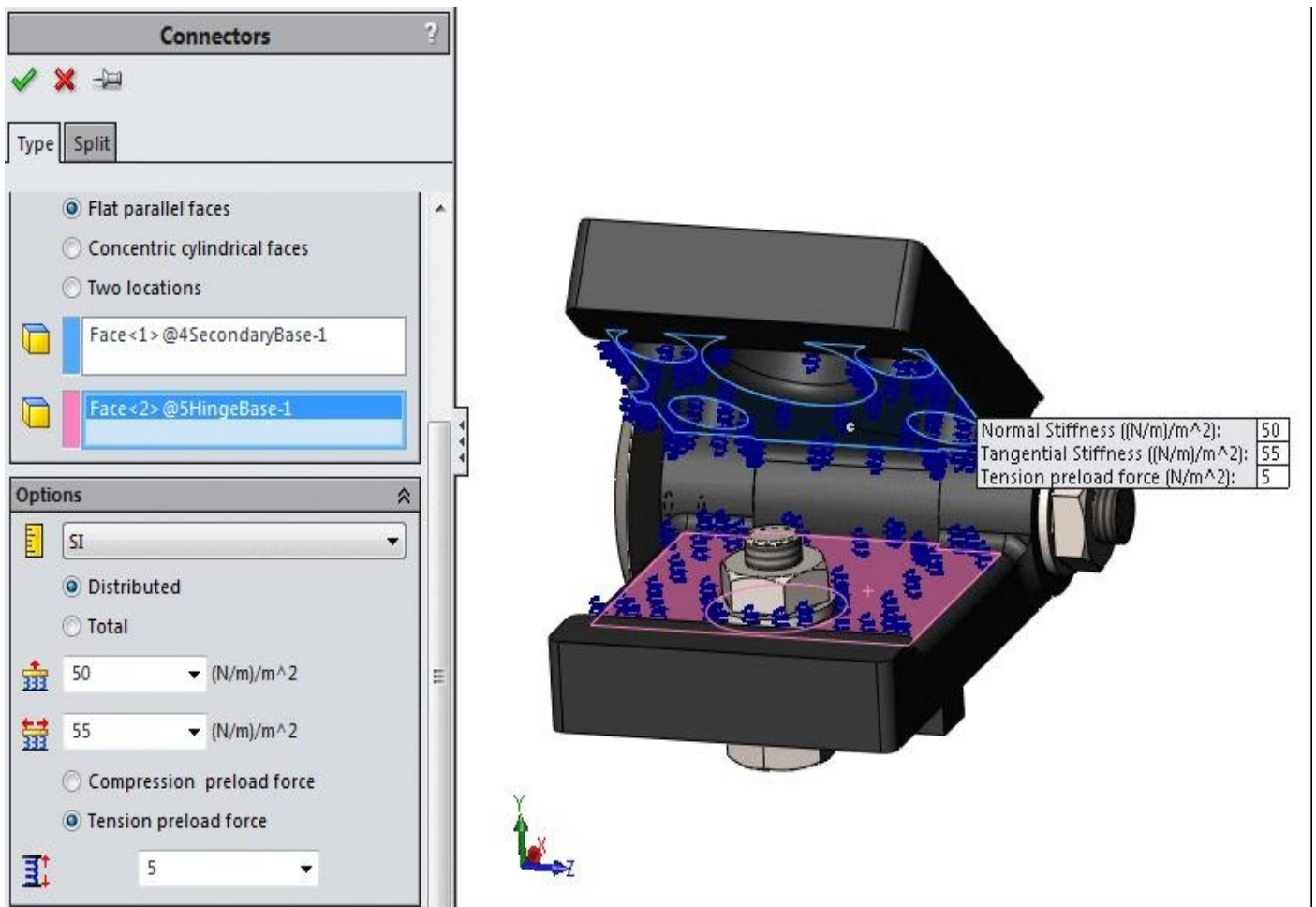


Figure-71. Preview of spring connector

- Click on the **OK** button from the **PropertyManager** to create the spring connection.

Pin

This option is used to connect two cylindrical components with the help of their circular faces. The assemblies that consist of multiple parts can be connected by the pin connector. Examples of assemblies with pins include laptop flap joined with base, scissors lifts, pliers, and actuators. The steps to use this option are given next.

- Click on the **Pin** button from the **Connections Advisor** drop-down. The **Connectors PropertyManager** will be displayed with the options related to pin connector; refer to [Figure-72](#).

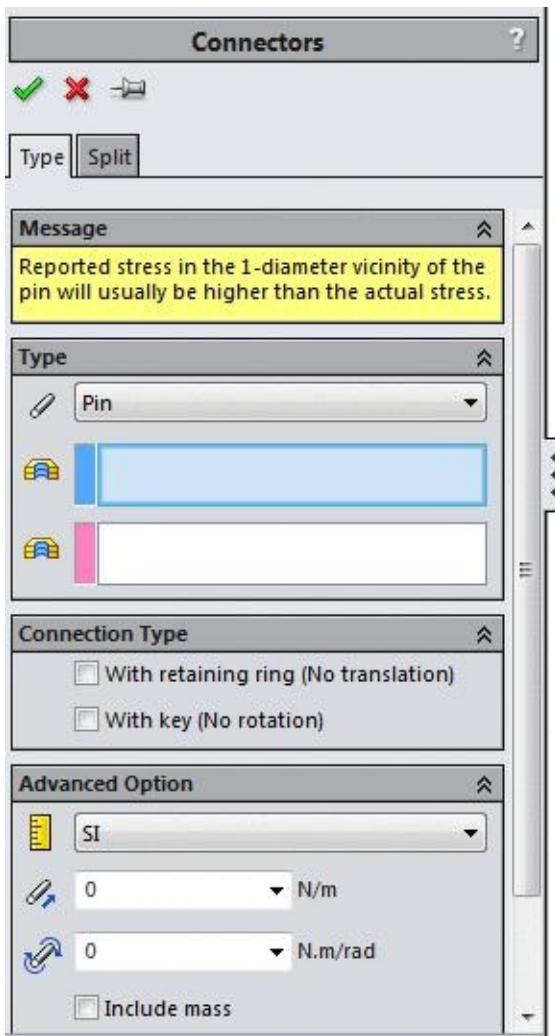


Figure-72. Connectors PropertyManager with Pin option

- Select the round face of the first object in assembly.
- Click in the pink selection box and select the round face of the other object in the assembly.
- Select the desired check boxes from the **Connection Type** rollout.
- Specify the relevant parameters in the **Advanced Option** rollout.
- Click on the **Include Mass** check box to include the mass of the pin. Specify the mass of pin in the edit box displayed.
- Click on the **Strength Data** check box and select the desired material by selecting the **Select material** button.
- Click on the **OK** button from the **PropertyManager** to apply the connection.

Bolt

This option is used to simulate the bolt connection between two components of the assembly. If you have added the bolts in the assembly by using the Toolbox then you can directly convert the bolt into simulation component by using **Yes** option from the dialog

box displayed on selecting the **Bolt** option from the **Connections Advisor** drop-down. The steps to use this option are given next.

- Click on the **Bolt** option from the **Connections Advisor** drop-down. If you have added the bolts in the assembly by using the **Toolbox**. Then the **Simulation** dialog box will be displayed as shown in [Figure-73](#). Click on the **Yes** button to automatically apply the simulation connection of bolt.

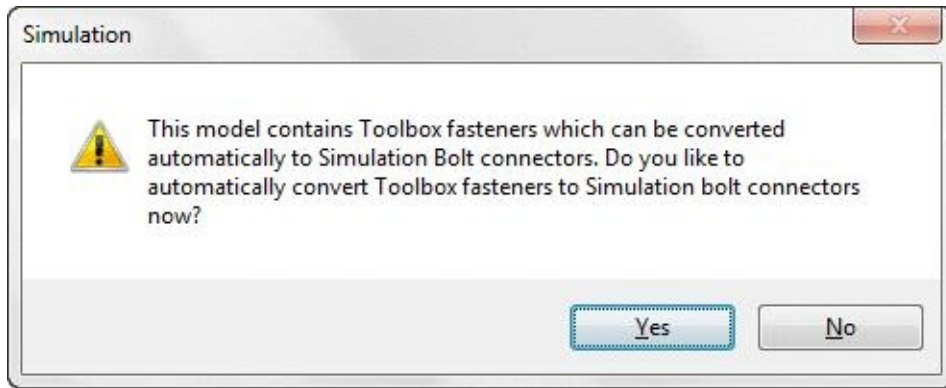


Figure-73. Simulation dialog box

- If the bolt are not taken from the **Toolbox** or you select the **No** button from the **Simulation** dialog box then the **Connectors PropertyManager** will display with the options related bolt connection; refer to [Figure-74](#).

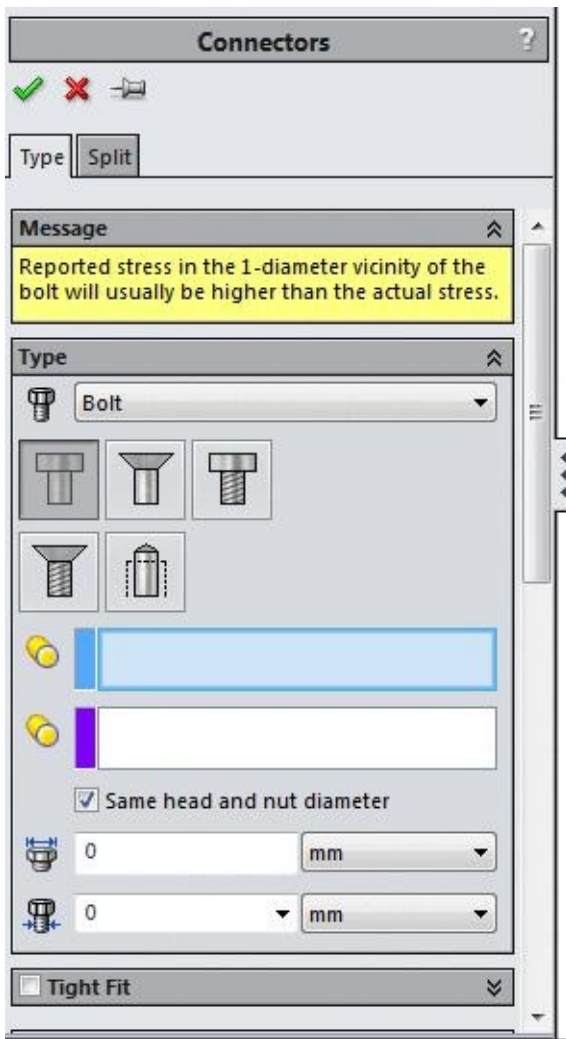


Figure-74. Connectors PropertyManager with Bolt option

- Select the desired button from the **Type** rollout to specify the type of bolt.
- Select the round edge of the bolt head.
- Click in the next selection box in the rollout and select the round edge of the nut. Preview of the bolt connector will be displayed; refer to [Figure-75](#).

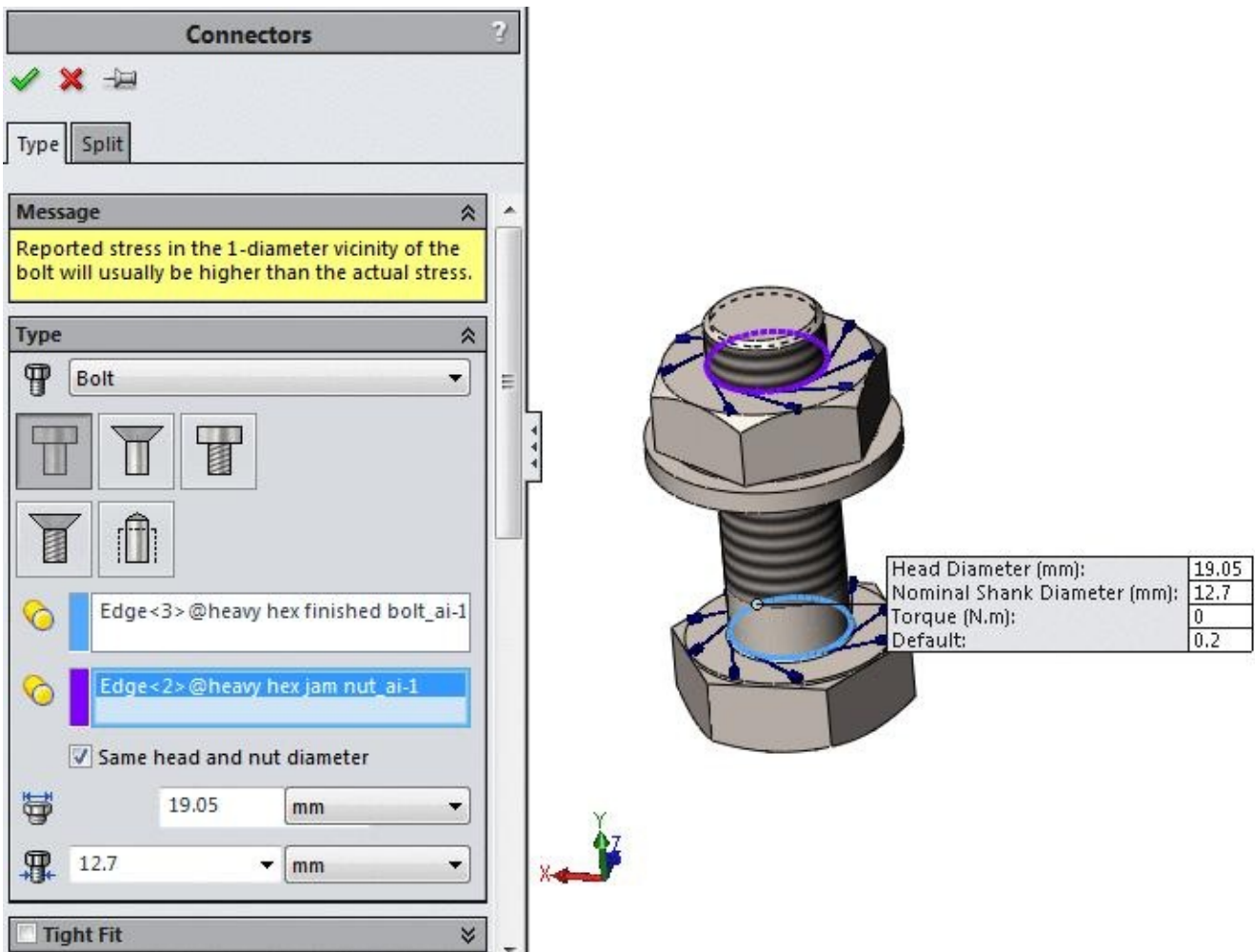


Figure-75. Preview of bot connector after selection

- Click on the **Tight Fit** check box to select the face with which the shank is in continuous contact.
- You can specify the material of the bolt by using the **Select Material** button available in the **Material** rollout. Then select the desired material to be applied for the connections.
- Click on the **Strength Data** check box and specify the related parameters using the options given in the rollout displayed.
- You can apply preload on the bolts to simulation the effect of tightened nuts by using the options in the Pre-load rollout.
- After specifying the desired parameters click on the **OK** button from the **PropertyManager** to apply the connection.

Spot Weld

This option is used to simulate the welding connection between two components at a specific points. The procedure to use this option are given next.

- Click on the **Spot Welds** option from the **Connections Advisors** drop-down. The **Connectors PropertyManager** will be displayed as shown in [Figure-76](#).

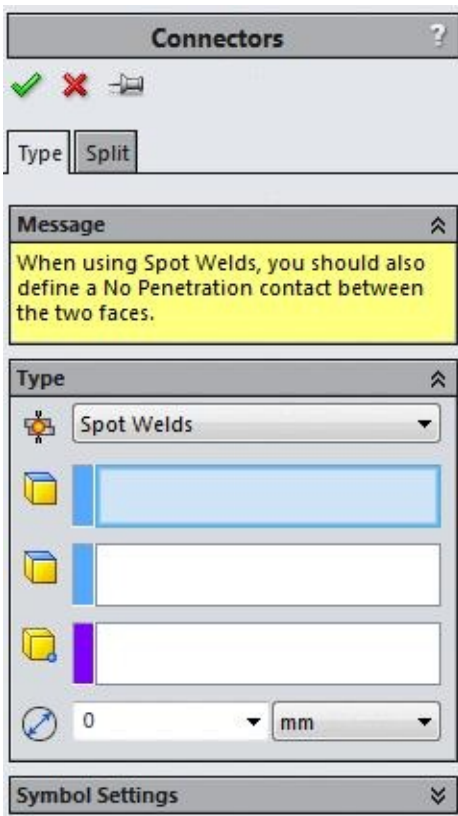


Figure-76. Connectors PropertyManager with Spot Welds option

- Select the first face that you want to be welded.
- Click in the next selection box in **PropertyManager** and select the second face.
- Now, click in the violet selection box and select the point at which both the plates meet each other.
- Click in the **Spot Weld Diameter** edit box and specify the diameter of the spot weld in the unit selected in drop-down.
- Click on the **OK** button from the **PropertyManager** to create the weld connection.

Edge Weld

This option is used to simulate the welding connection between two components at a common edge. The procedure to use this option is given next.

- Click on the **Edge Weld** button from the **Connections Advisor** drop-down. The **Edge Weld Connector PropertyManager** will be displayed as shown in [Figure-77](#).

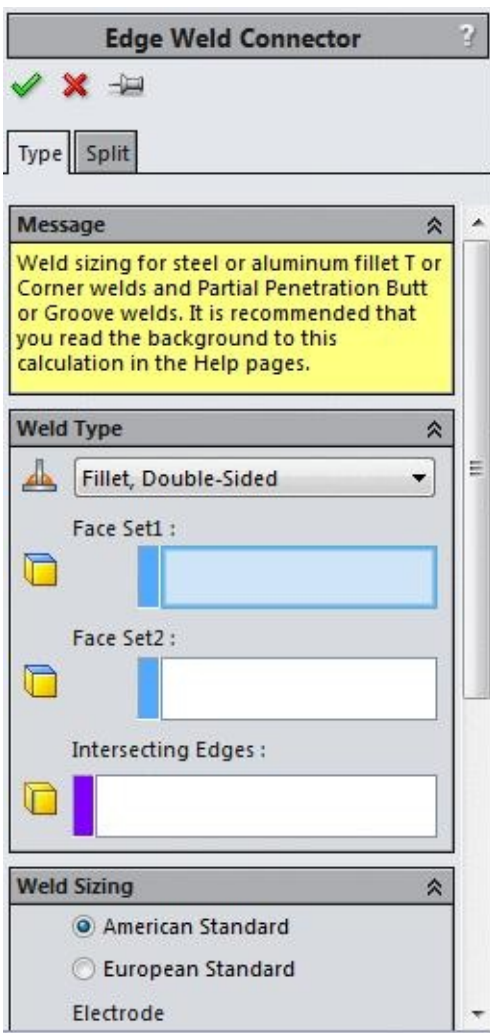


Figure-77. Edge Weld Connector PropertyManager

- Click on the first drop-down list in the **Weld Type** rollout in **PropertyManager** and select the type of fillet/groove you want to generate on the welding bead.
- Select the first face for welding. Note that you can select only the surface or sheet metal components to apply this connection.
- Click in the next selection box and select the second face for welding.
- Click in the **Intersecting Edges** selection box and select the edge at which the selected two faces intersect.
- You can set the size of welding by using the options in the **Weld Sizing** rollout. You can also switch between American standard and European standard of weld sizing by selecting the corresponding radio button.

Link

This option is used to simulate the connection between two components like the two components are hinged at their specified points. The procedure to use this option is given next.

- Click on the **Link** button from the **Connections Advisor** drop-down. The **Connectors PropertyManager** will be displayed as shown in [Figure-78](#).

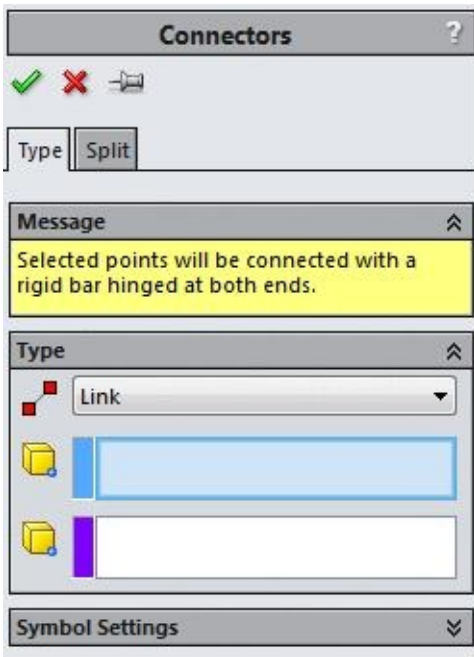


Figure-78. Connectors PropertyManager with Link option

- Click on the desired point of first entity.
- Click in the violet colored selection box and select the second point of link on the second entity.
- Click on the **OK** button to create the link. Note that on specifying the link connector, the distance between the two selected points will remain the same.

Bearing

This option is used to simulate the connection between two components like the two components are linked to each other with the help of a bearing. This connection is generally applied to shafts and housings. The procedure to use this option is given next.

- Click on the **Bearing** button from the **Connections Advisor** drop-down. The **Connectors PropertyManager** will be displayed with the options related to bearing connection; refer to [Figure-79](#).

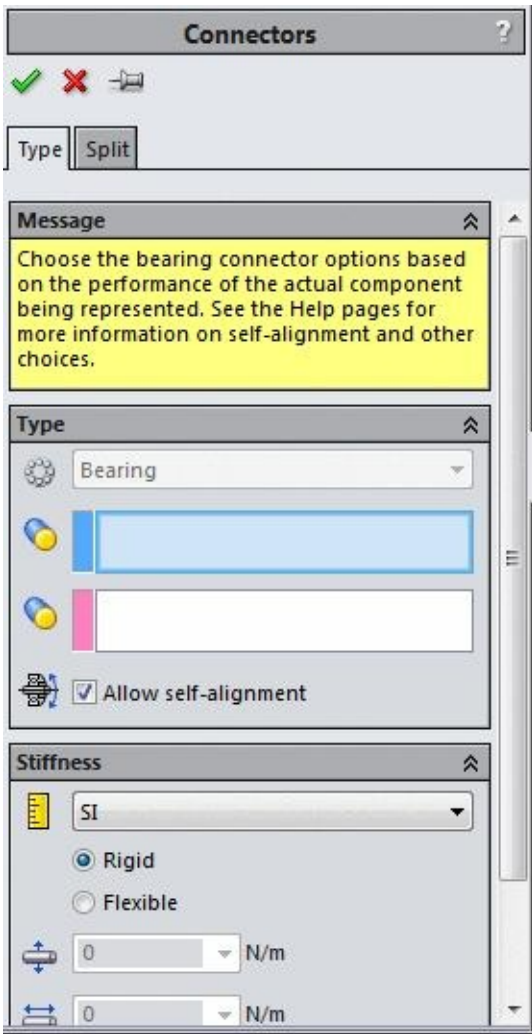


Figure-79. Connectors PropertyManager with bearing option

- Select the round face of the shaft.
- Click in the next selection box in the **Type** rollout and select the round face of the housing.
- Specify the desired parameters in the **Stiffness** rollout and click on the **OK** button from the **PropertyManager** to apply the connection. See [Figure-80](#).

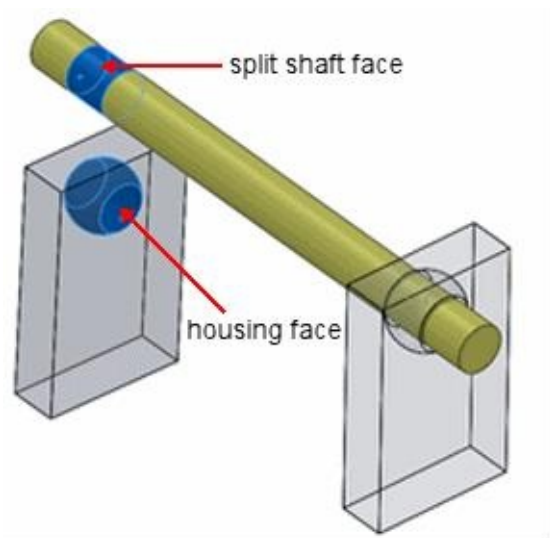


Figure-80. Preview of bearing connection

Rigid Connection

This option is used to apply rigid connection between the two components. The procedure to create the rigid connection is given next.

- Click on the **Rigid Connection** button from the **Connections Advisor** drop-down. The **Connectors PropertyManager** will be displayed as shown in [Figure-81](#).

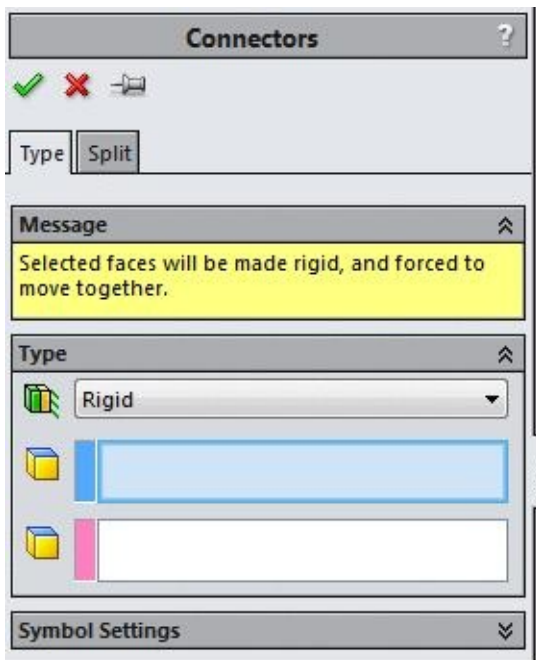


Figure-81. Connectors PropertyManager with rigid option

- Select the first face for applying rigid connection.
- Click in the next selection box and select the second adjoining face.
- Click on the **OK** button from the **PropertyManager** to create the connection.

After specifying the desired connections for the assembly/component, the next step is to create and refine mesh for the assembly/component. The methods to create mesh are discussed next.

MESHING

Meshing is the base of FEM. Meshing divides the solid/shell models into elements of finite size and shape. These elements are joined at some common points call nodes. These nodes define the load transfer from one element to other element. Meshing is a very crucial step in design analysis. The automatic mesher in the software 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.

The software 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 generates 3D tetrahedral solid elements, 2D triangular shell elements, and 1D beam elements. A mesh consists of one type of elements unless the mixed mesh type is specified. Solid elements are naturally suitable for bulky models. Shell elements are naturally suitable for modeling thin parts (sheet metals), and beams and trusses are suitable for modeling structural members.

The procedure to create the mesh of the solid is given next.

- Click on the **Create Mesh** button from the **Run** drop-down in the **Ribbon**. The **Mesh PropertyManager** will be displayed as shown in [Figure-82](#).

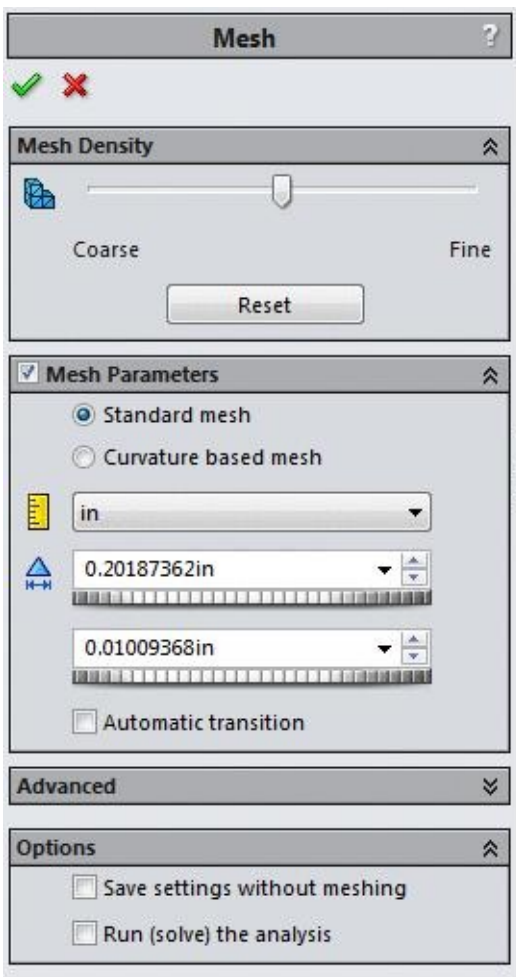


Figure-82. Mesh PropertyManager

- Move the slider to make the elements of mesh fine or coarse. If you move the slider towards the right, the elements of the mesh will become fine. If you move the slider towards the left, the mesh will become coarse.
- Select the **Mesh Parameters** check box to specify the parameters for mesh.
- Click on the **Curvature based mesh** radio button if you want to create a curve based mesh.
- Select the **Automatic transition** to apply mesh controls to small features, holes, fillets, and other fine details of your model. Clear **Automatic transition** before meshing large models with many small features and details to avoid generating large numbers of elements.
- Click on the **Advanced** rollout to expand the advanced options related to the mesh.
- Jacobian points are used to specify the points on which the distortion will be created while performing the analysis. To set the Jacobian points, click on the drop-down in the **Advanced** rollout and select the desired number of points.
- Select the Draft Quality Mesh check box to set the quality of mesh for fast evaluation of the analysis. If you select this check box then 4 corner nodes for solid and 3 corner nodes for each shell element are created.
- **Automatic trails for solid** check box is selected to find out and apply the optimum mesh size automatically. Note that the ratio by which the global element size and tolerance are reduced for each trial is **0.8**. On selecting this check box, the **Number of trails** spinner is displayed. Using this spinner, you can specify the maximum number of trials for re-meshing.
- If your part fails while meshing at some portions, then select the **Remesh failed parts with incompatible mesh** check box to create incompatible mesh at the portions that failed.
- If you do not want to actually mesh but want to save the settings then select the **Save settings without meshing** check box.
- To run the analysis directly after creation of mesh, select the **Run (solve) the analysis** check box.

[Figure-83](#) shows the meshing of a model with the loads and fixtures applied.

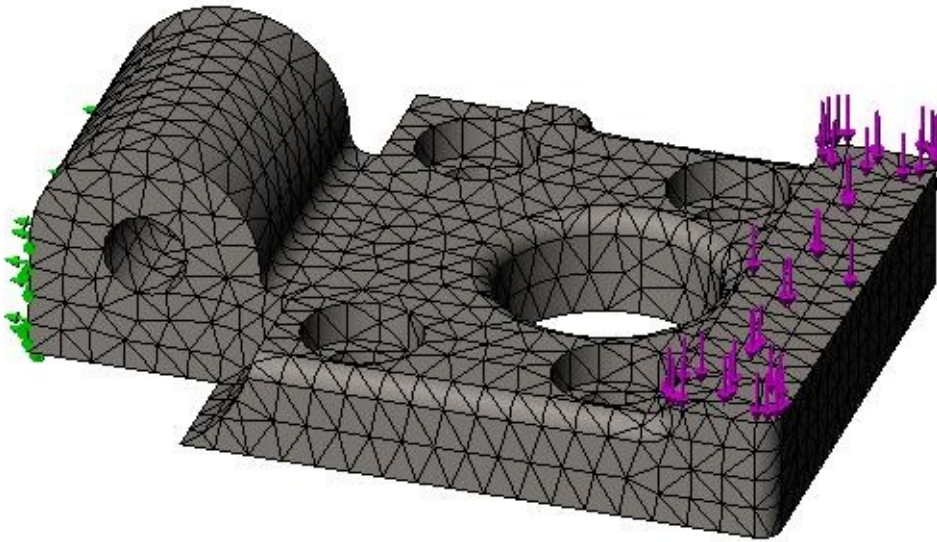


Figure-83. Preview of mesh with loads applied

RUNNING ANALYSIS

After we have applied all the parameters related to analysis and we have created the mesh, the next step is to run the analysis. The steps to run the analysis are given next.

- Click on the **Run** button from the **Ribbon**. The solving will start and the result will be displayed; refer to [Figure-84](#).

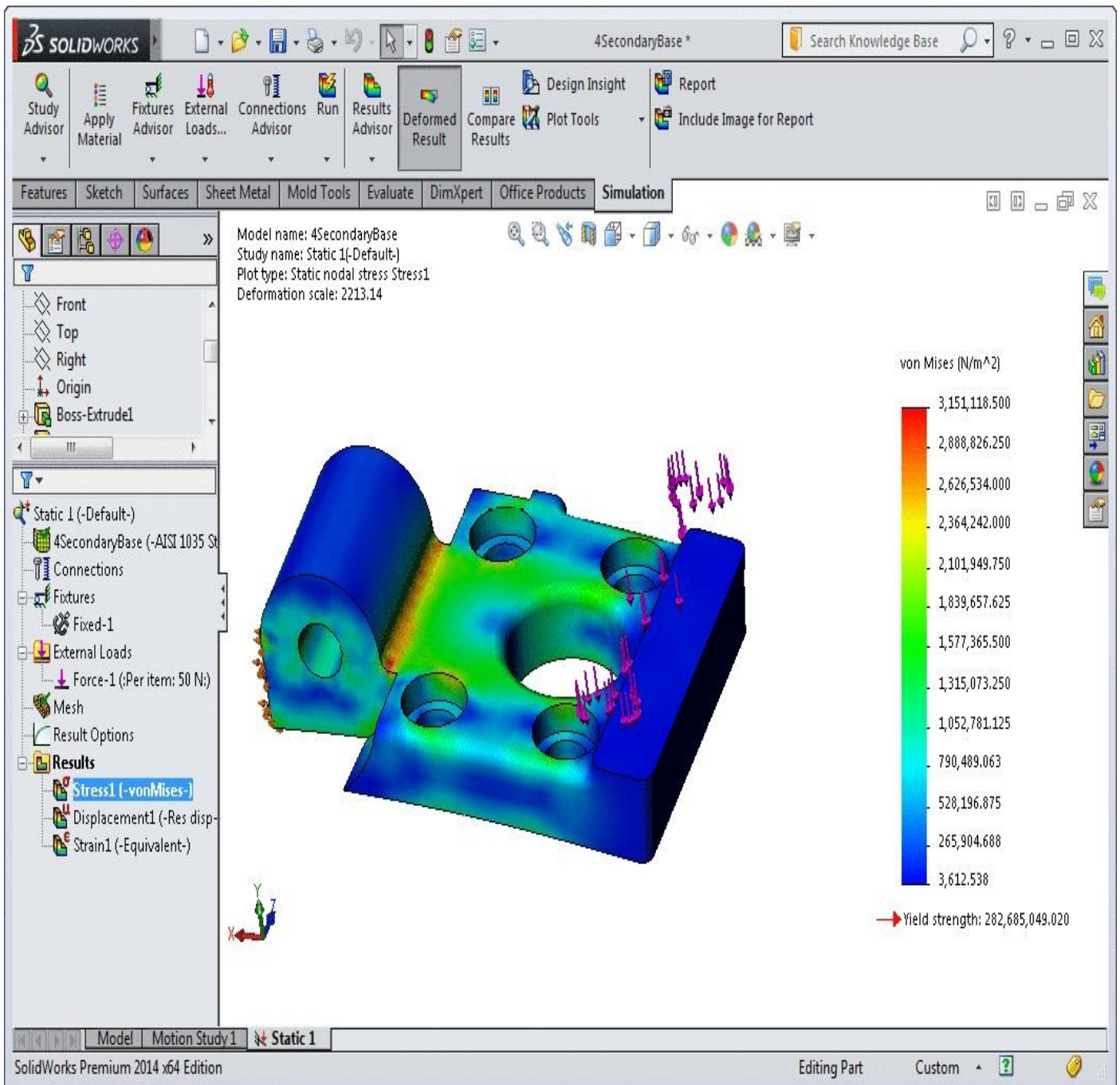


Figure-84. Preview of simulation result

- After we run the analysis, the next step is to interpret the results of the analysis. To interpret the results for various purposes, the tools are available in the **Result Advisors** drop-down which is discussed next.

RESULTS ADVISOR

The tools in this drop-down are available only after you successfully run the desired type of analysis. The tools that get available after running static analysis are given next.

- Results Advisor

- Model Only (No Results)
- New Plot menu
- List Stress, Displacement and Strain
- List Result Force
- List Contact Force
- List Pin\Bolt\Bearing Force

The tools displayed above are discussed next.

Results Advisor

This tool is used to display the result page of the **Simulation Advisor**; refer to [Figure-85](#). The steps to use the options in this page of Simulation Advisor are given next.

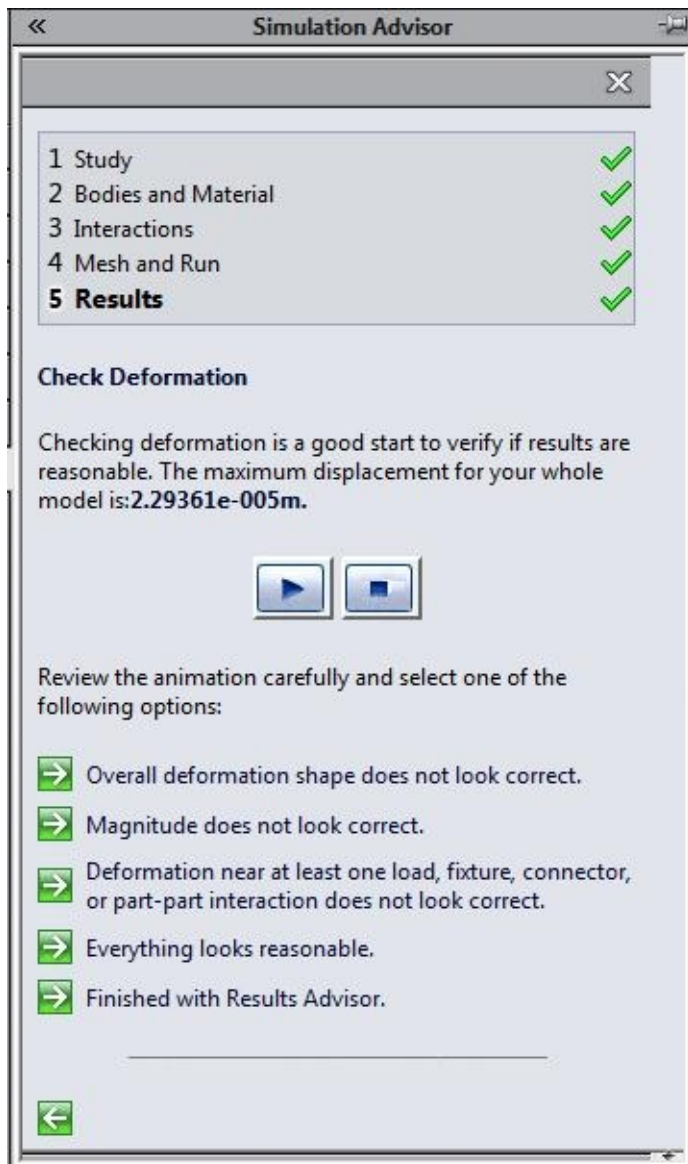


Figure-85. Results page Simulation Advisor

- Click on the **Play** button from the **Simulation Advisor** to plot the simulation of the applied analysis. Note that the limits of deflection are displayed in the **Simulation Advisor**. For example, in [Figure-85](#), the maximum deflection is **2.29361e-005m**.
- After you have checked the simulation, you need to change the parameters to get the desired results. For example, you want to change the material to increase the stiffness of the model. To preform these changes, we are provided with the three link buttons in the Simulation Advisor.
- Click on the desired link button from the first three link buttons to change the parameters. We will discuss these options later in the book.

Model Only(No Results)

This tool is used to display the model without displaying any result of analysis. Use of this tools is very simple, just click on the tool and the analysis result will hid automatically.

New Plot

The options in this cascading menu are used generate new plots for various parameters on the basis of performed analysis. In case of a Static analysis, you can get the options as shown in [Figure-86](#).

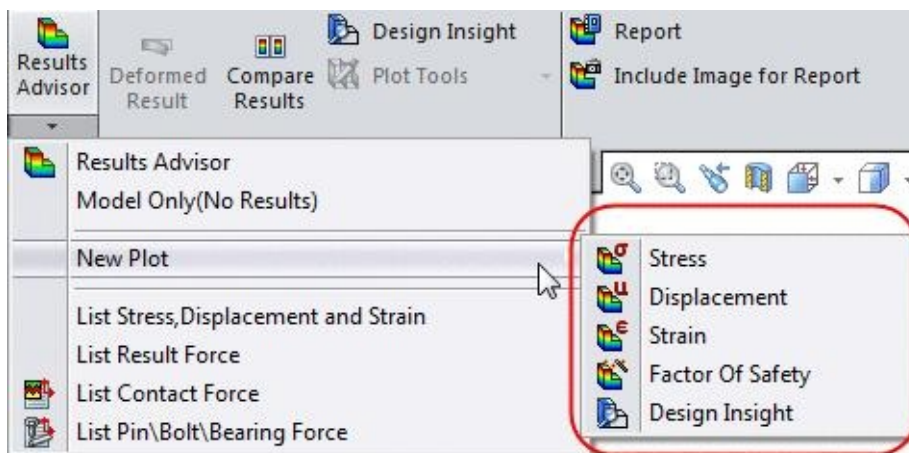


Figure-86. Options in New Plot cascading menu

The procedure to use these options is given next.

- Click on the desired option from the menu. The respective **PropertyManager** will display; refer to [Figure-87](#) for **Design Insight**.

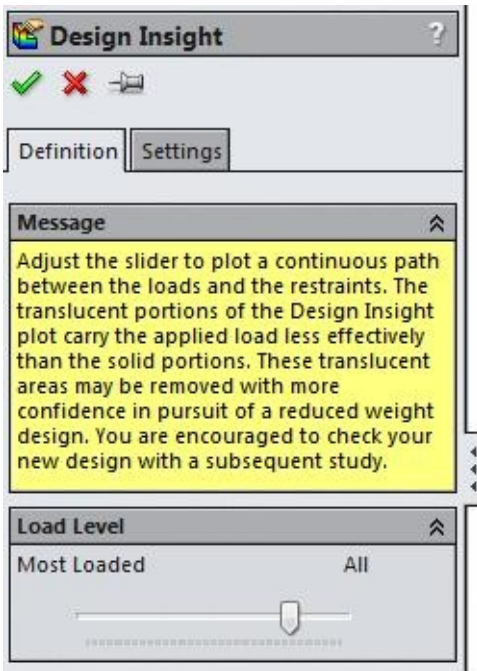


Figure-87. Design Insight PropertyManager

- Specify the desired settings in the **PropertyManager** and click on the **OK** button display the desired type of analysis result; refer to [Figure-88](#). Note that the link for selected analysis result will be added in the **Results** node in the left area.

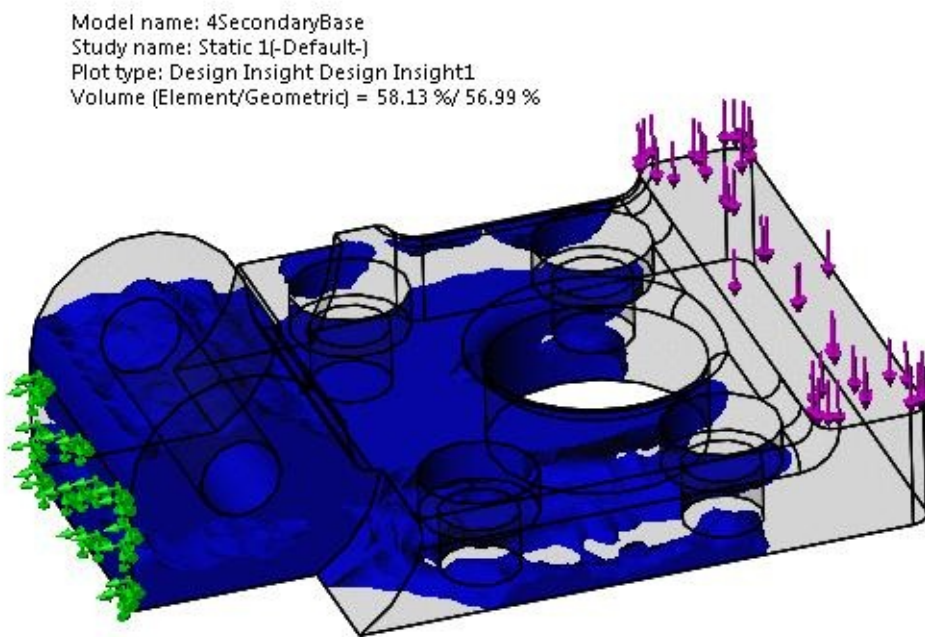


Figure-88. Preview for design insight

List Stress, Displacement and Strain

This tool is used to display the values of stress, displacement and strain at node intersecting with the selected plane. The procedure to use this option is given next.

- Select the plane by using which you want to check values of stress, displacement and strain at the intersecting nodes.
- Click on the **List Stress, Displacement and Strain** tool from the **Results Advisor** drop-down. The **List Results PropertyManager** will be displayed as shown in [Figure-89](#).

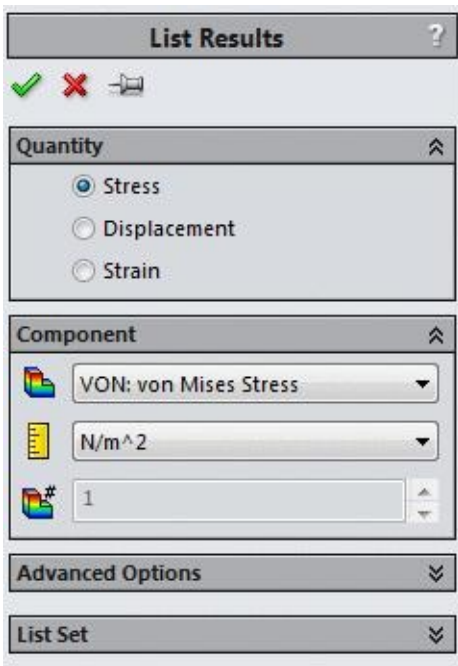


Figure-89. List Results PropertyManager

- Select the desired radio button from the **Quantity** rollout. The options related to the selected radio button will be displayed in the **Component** rollout. Select the desired component of stress/displacement/strain from the first drop-down in the rollout.
- Click on the **Advanced Options** to expand the rollout. The advanced options related to the selected radio button will be displayed. Refer to [Figure-90](#).

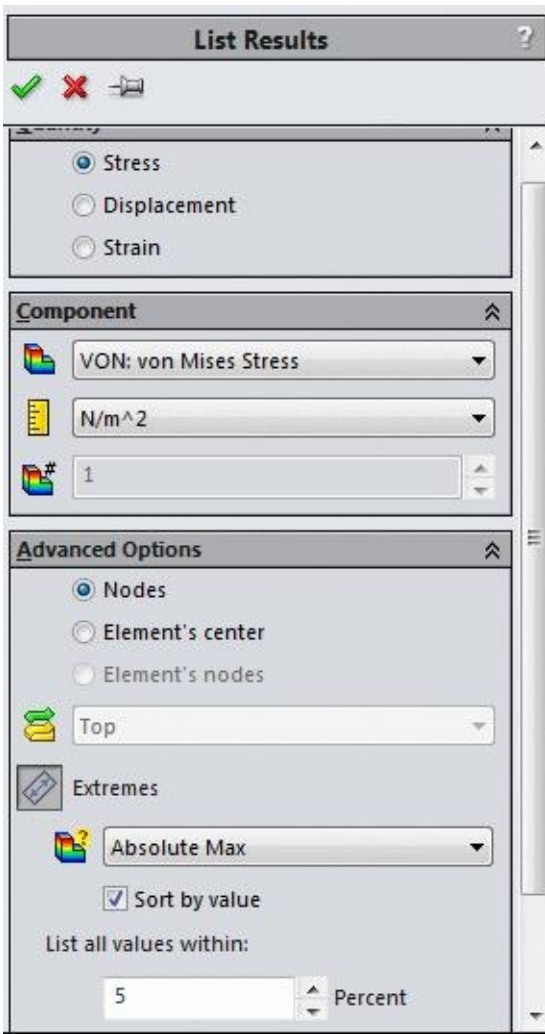


Figure-90. List Results PropertyManager with expanded Advanced Options rollout

- Select the desired parameters from the **Advanced Options** rollout to set the limits of displayed result.
- Click on the **OK** button from the **PropertyManager**. The **List Results** dialog box will be displayed; refer to [Figure-91](#).

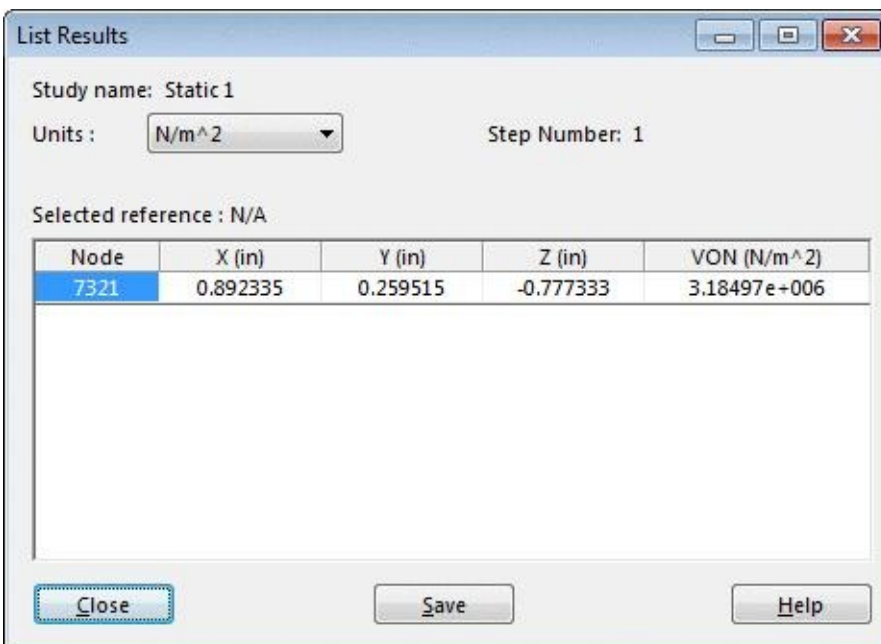


Figure-91. List Results dialog box

- Click on the **Save** button to save the results in a CSV excel file. Click on the **Close** button to close the dialog box.

Result Force

This tool is used to display the reaction forces resulting due to the applied load. The steps to use this tool are given next.

- Click on the **Result Force** tool from the **Results Advisor** drop-down. The **Result Force PropertyManager** will be displayed as shown in [Figure-92](#).
- Select the desired radio button from the **PropertyManager**.
- Click in the blue selection box and select the face/edge/point at which you want to check the reaction forces.
- Select the desired unit system from the **Unit** drop-down in the **PropertyManager**.
- Click on the **Update** button from the **PropertyManager** to display the reaction forces. The box containing values of reaction forces will be displayed with the model; refer to [Figure-93](#).

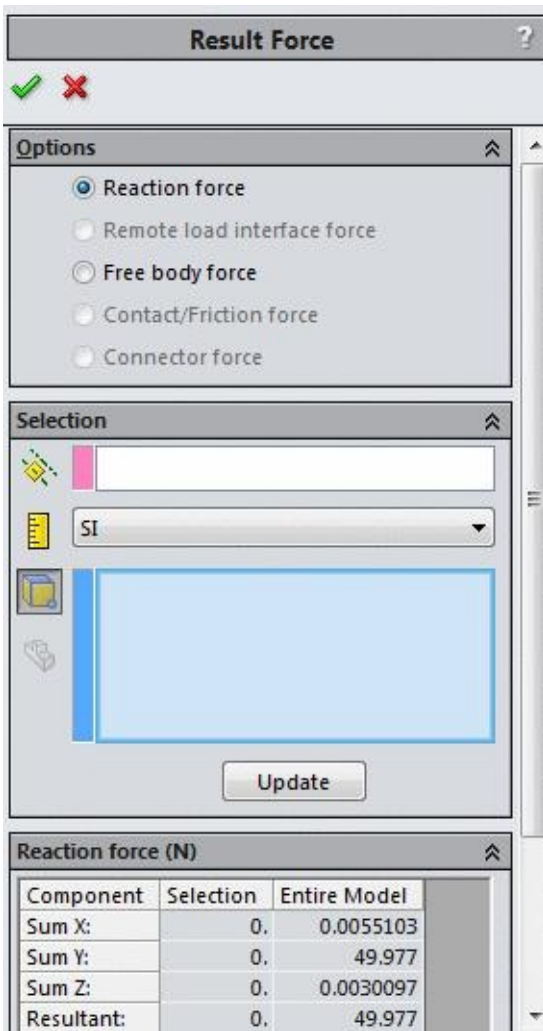


Figure-92. Result Force PropertyManager

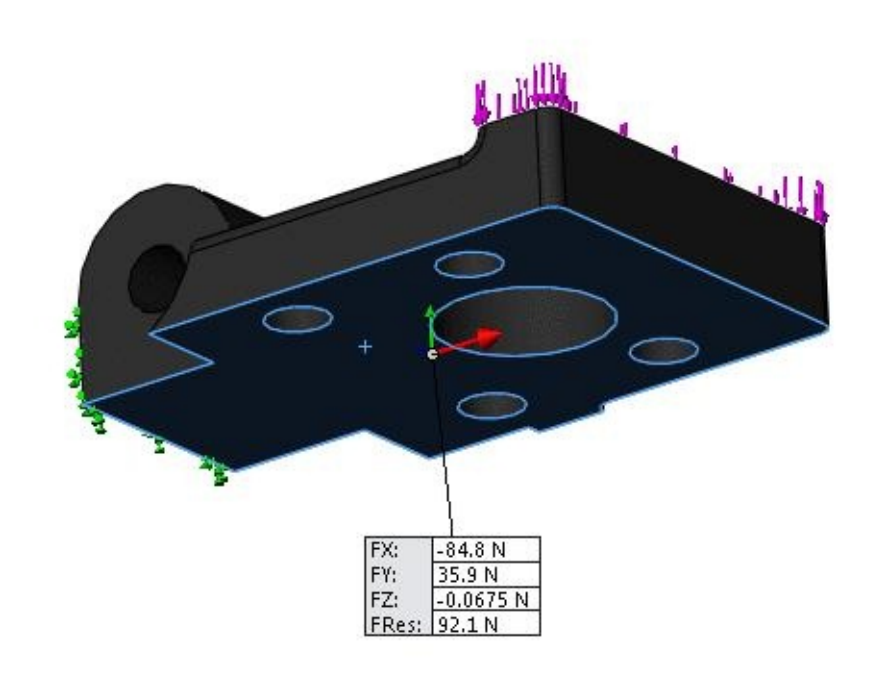


Figure-93. Preview of result force

- Click on the **OK** button from the **PropertyManager** to exit the tool.

If you have applied contact forces or Pin/Bolt/Bearing forces, then click on the respective tool from the Results Advisor drop-down. The result will be displayed in the same way as discussed earlier.

We will learn about more options related to results while performing the analyses.

STATIC ANALYSIS

CHAPTER 3



Topics Covered

The major topics covered in this chapter are:

- *Starting Static Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

LINEAR STATIC ANALYSIS

Linear static analysis is performed to calculate stresses, displacements, strains, forces, and error estimates under various loading conditions. On applying the loads, the body gets deformed and the loads are transmitted throughout the body.

Assumptions

Some assumptions are made in this type of analysis, like:

1. The loads applied does not vary with time,
2. All loads are applied slowly and gradually until they reach to the full magnitude. And after reaching the full magnitude, the loads remains constant. Thereby, neglecting impact, inertial, and damping forces.
3. The materials applied to the components satisfy the Hooke's law.
4. The change in stiffness due to loading is neglected.
5. Boundary conditions do not vary during the application of loads. Loads must be constant in magnitude, direction, and distribution.

Geometry Assumptions

1. The part model must represent the required CAD geometry.
2. Only the internal fillets in the area of interest will be included in the study.
3. Shells are created when thickness of the part is small in comparison to its width and length.
4. Thickness of the shell is assumed to be constant.
5. If the dimensions of a particular part are not critical and will not affect the analysis results, some approximations can be made in modeling the particular part.
6. Primary members of structure are long and thin such that a beam idealization is required.
7. Local behavior at the joints of beams or other discontinuities are not of primary interest such that no special modeling of these area is required.
8. Decorative or external features will be assumed insignificant to the stiffness and the performance of the part and will be omitted from the model.

Material Assumptions

1. Material remain in the linear regime. It is understood that

either stress levels exceeding yield or excessive displacements will constitute a component failure. That is non linear behavior cannot be accepted.

2. Nominal material properties adequately represent the physical system.
3. Material properties are not affected by load rate.
4. Material properties can be assumed isotropic (Orthotropic) and homogeneous.
5. Part is free of voids or surface imperfections that can produce stress risers and skew local results.
6. Actual non linear behavior of the system can be extrapolated from the linear material results.
7. Weld material and the heat affected zone will be assumed to have same material properties as the base material.
8. Temperature variations may have a significant impact on the properties of the materials used. Change in material properties is neglected.

Boundary Conditions Assumptions

1. Choosing proper BC's require experience.
2. Using BC's to represent parts and effects that are not or cannot be modeled leads to the assumption that the effects of these un-modeled entities can truly be simulated or has no effect on the model being analyzed.
3. For a given situation there would be many ways of applying boundary conditions. But these various alternatives can be wrong if the user does not understand the assumptions they represent.
4. Symmetry/ anti-symmetry/ reflective symmetry/ cyclic symmetry conditions if exists can be used to minimize the model size and complexity.
5. Displacements may be lower than they would have been had the boundary conditions been more appropriate. Stress magnitudes may be higher or lower depending on the constraint used.

Fasteners Assumptions

1. Residual stress due to fabrication , pre-loading of bolts, welding and/or other manufacturing or assembly processes are neglected.
2. Bolt loading is primarily axial in nature.
3. Bolt head or washer surface torque loading is primarily axial in

nature.

4. Surface torque loading due to friction will produce only local effects.
5. Bolts, spot welds, welds, rivets, and/or fasteners which connect two components are considered perfect and acts as rigid joint.
6. Stress relaxation of fasteners or other assembly components will not be considered. Load on threaded portion of the part is evenly distributed on engaged threads
7. Failure of fasteners will not be reflected in the analysis.

General Assumptions

1. If the results in the particular area are of interest, then mesh convergence will be limited to this area.
2. No slippage between interfacing components will be assumed.
3. Any sliding contact interfaces will be assumed frictionless.
4. System damping will be normally small and assumed constant across all frequencies of interest unless otherwise available from published literature or actual tests.
5. Stiffness of bearings in radial or axial directions will be considered infinite.
6. Elements with poor or less than optimal geometry are only allowed in areas that are not of concern and do not affect the overall performance of the model.

When a system under load is analyzed with linear static analysis, the linear finite element equilibrium equations are solved to calculate the displacement components at all nodes. These results are used to calculate the strain components. These strain results along with stress-strain relationship helps to calculate stresses.

The procedure given next is generic and can be applied on any model/assembly. But, we will use a model to help you understand the complete procedure with real world problems.

PERFORMING STATIC ANALYSIS

To perform the static analysis on any model, make sure that you have opened the model file in SolidWorks. The file used in this example can be found in the resource kit for this book. After opening the file follow the steps given next.

Starting the Static Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. Refer to [Figure-1](#). List of analysis studies that can be performed, will be displayed.

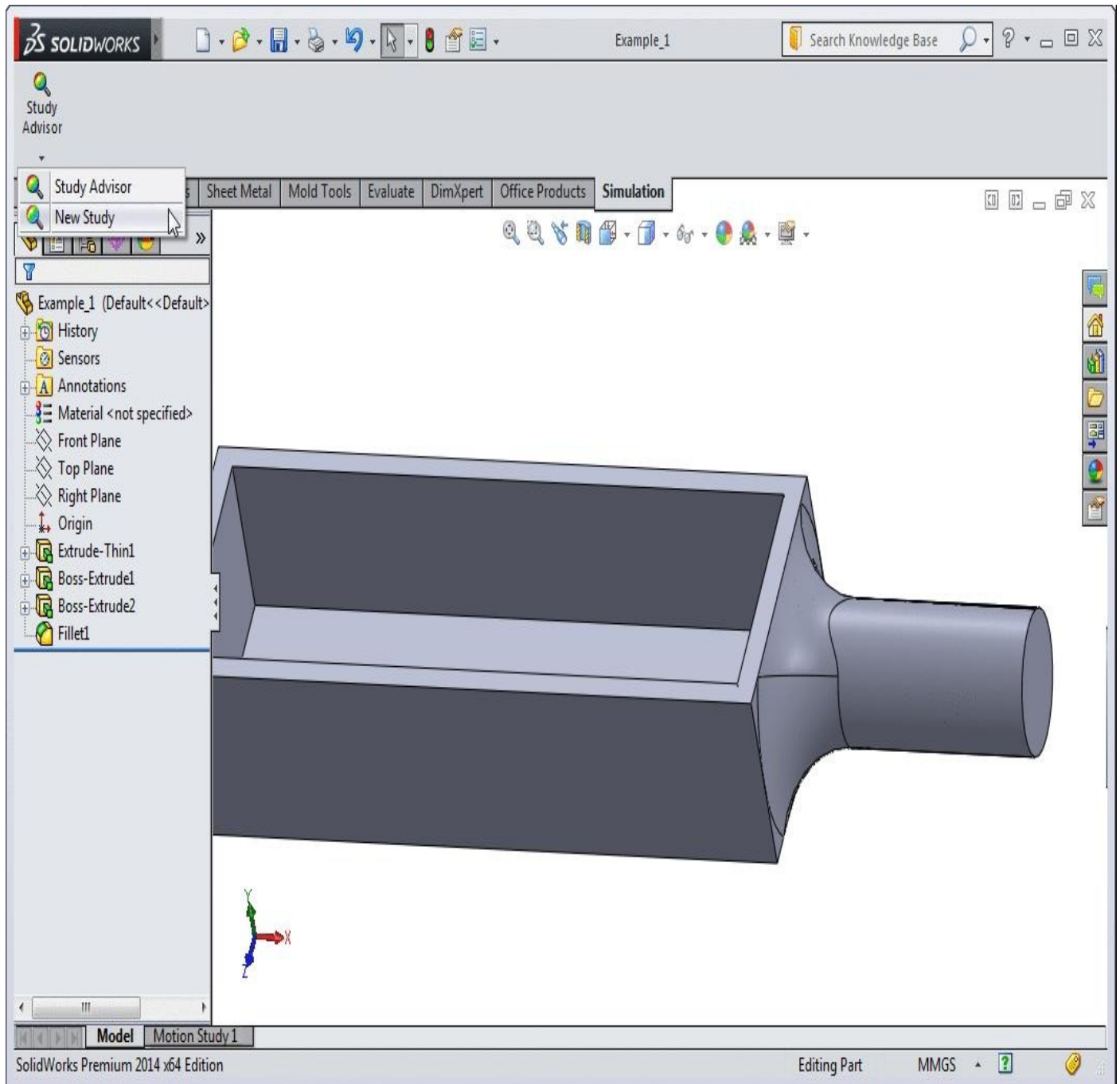


Figure-1. Starting Analysis

- Click on the **Static** button if not selected. Specify the name of the analysis in the edit box displayed above the list of analyses.

- Click on the **OK** button from the **PropertyManager**. **Static 1** tab will be added at the bottom of the modeling area in the **StudyBar**. Also the tools related to the analysis will be displayed in the **Ribbon**; refer to [Figure-2](#).

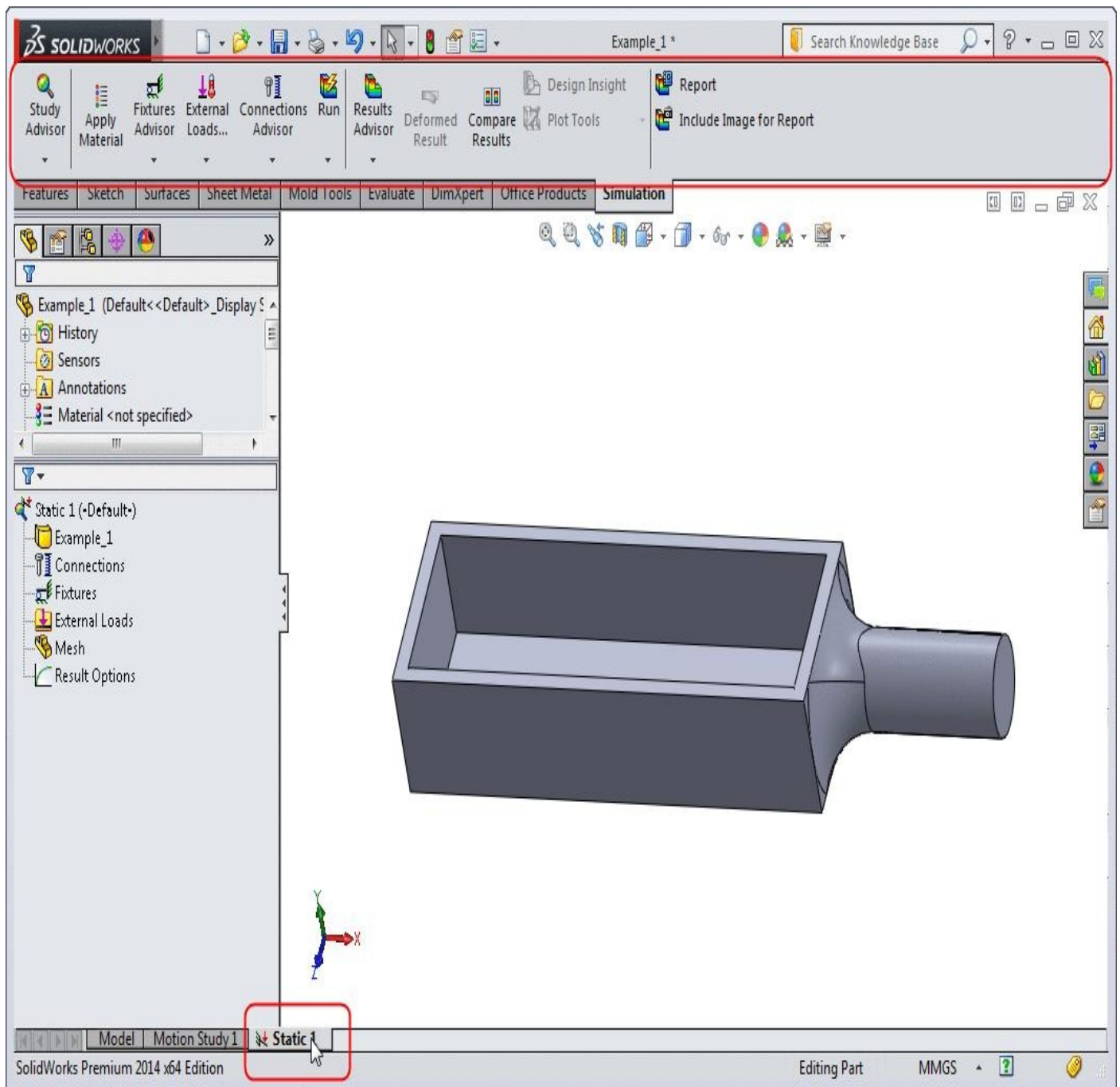


Figure-2. Analysis environment

Applying Material

- Click on the **Apply Material** button from the **PropertyManager**. The **Material** dialog box will be displayed; refer to [Figure-3](#).

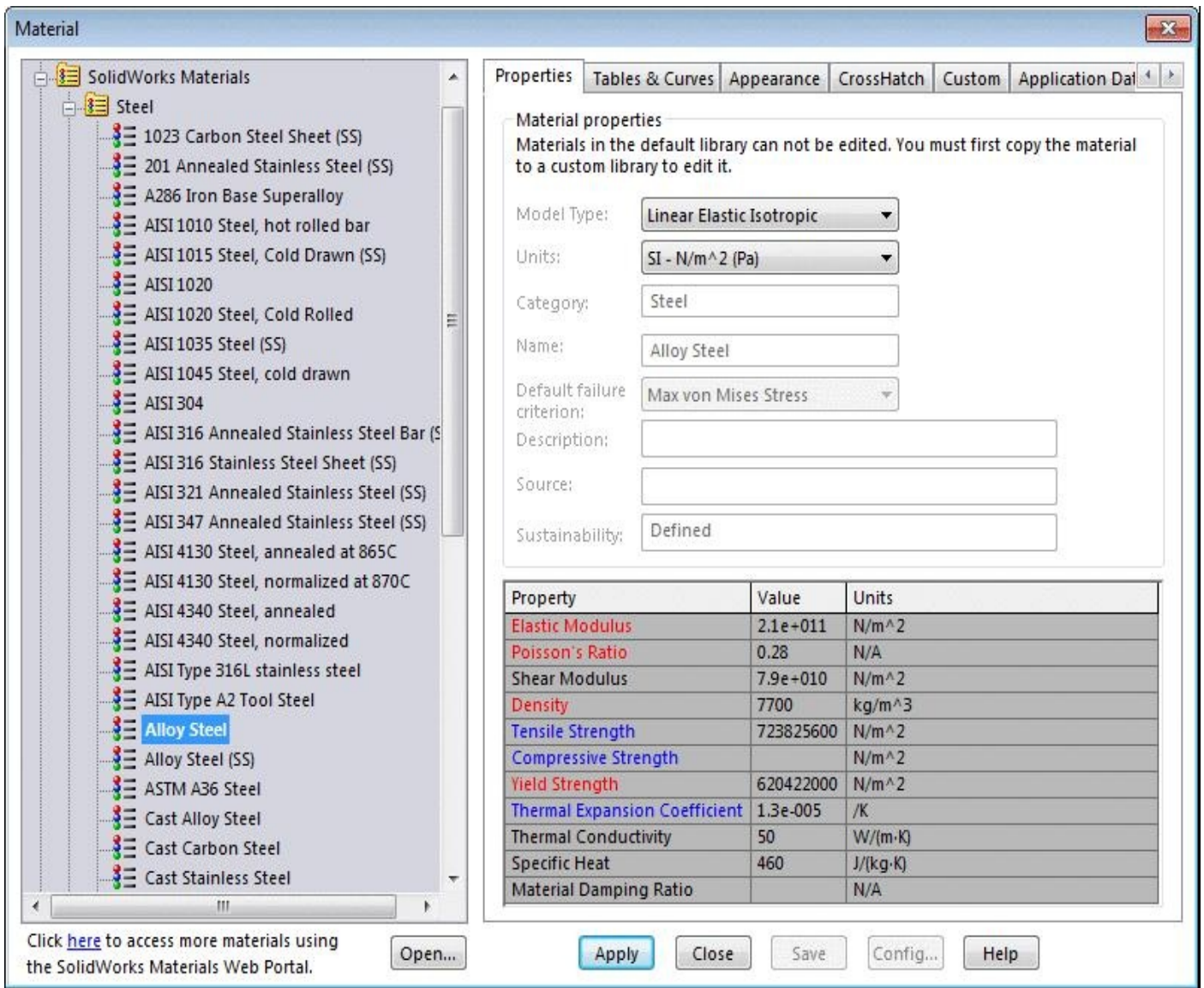


Figure-3. Material dialog box

- Select the **Cast Alloy Steel** from the left area of the dialog box and select the **Apply** button to apply the material.
- Click on the **Close** button to close the dialog box.

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed; refer to [Figure-4](#).

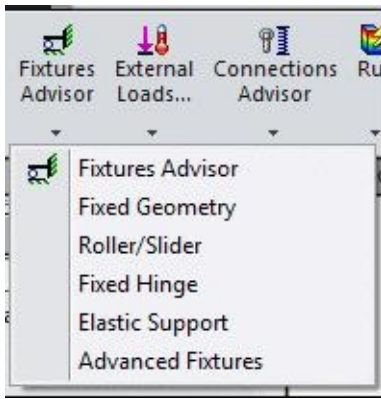


Figure-4. Fixtures Advisor

- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed; refer to [Figure-5](#).

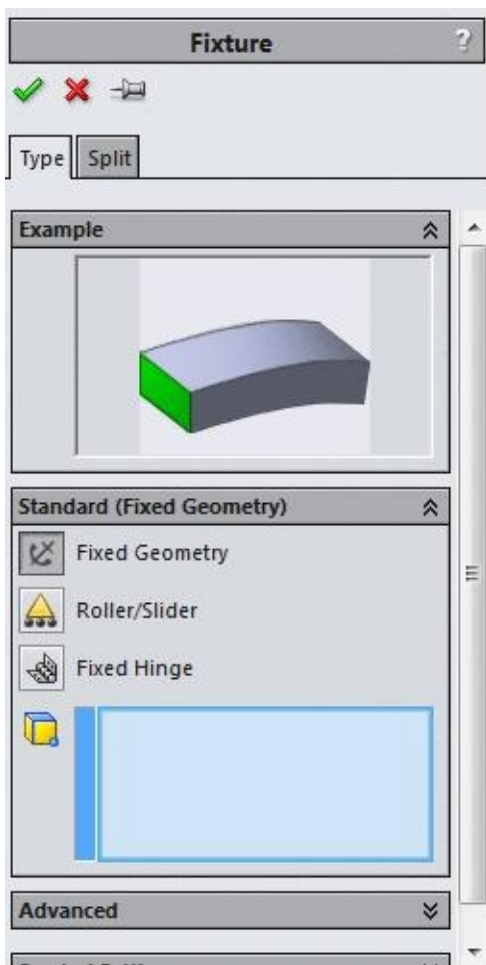


Figure-5. Fixture PropertyManager

- Select the round face of the model to make it fixed; refer to [Figure-6](#).

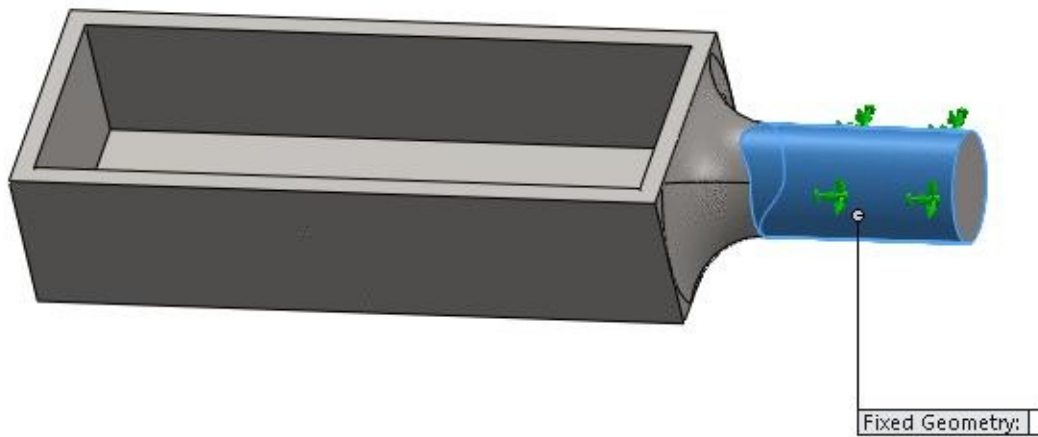


Figure-6. Fixed geometry

- Click on the **OK** button from the **PropertyManager** to fix the round face.

Now, we need to specify the loads that are actually occurring on the component.

Applying Loads

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed; refer to [Figure-7](#).

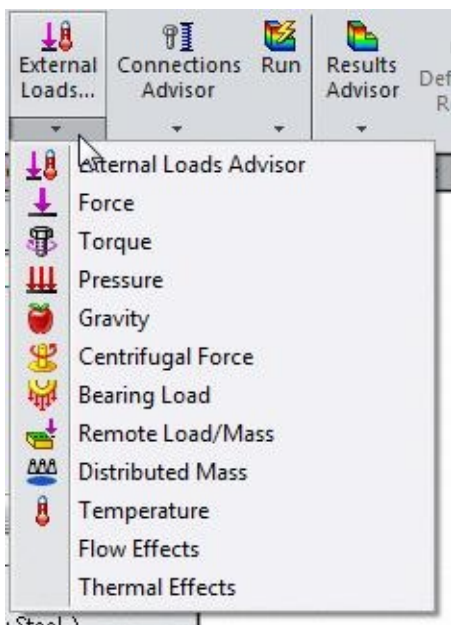
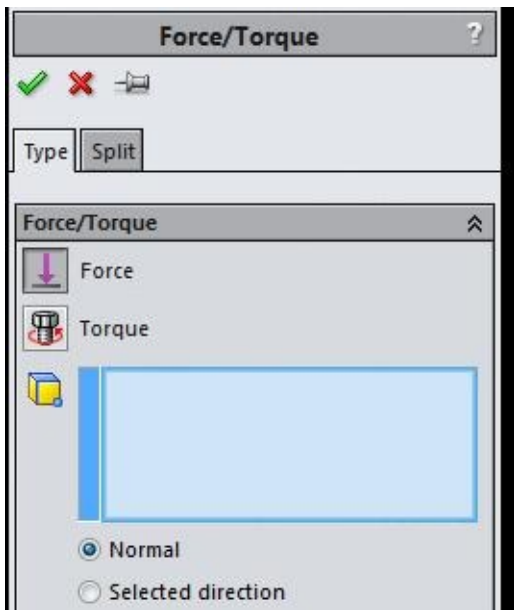


Figure-7. External loads advisor

- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed as shown in [Figure-8](#).

Figure-8. Force Torque PropertyManager



- Select the face as shown in [Figure-9](#) to apply the force load.

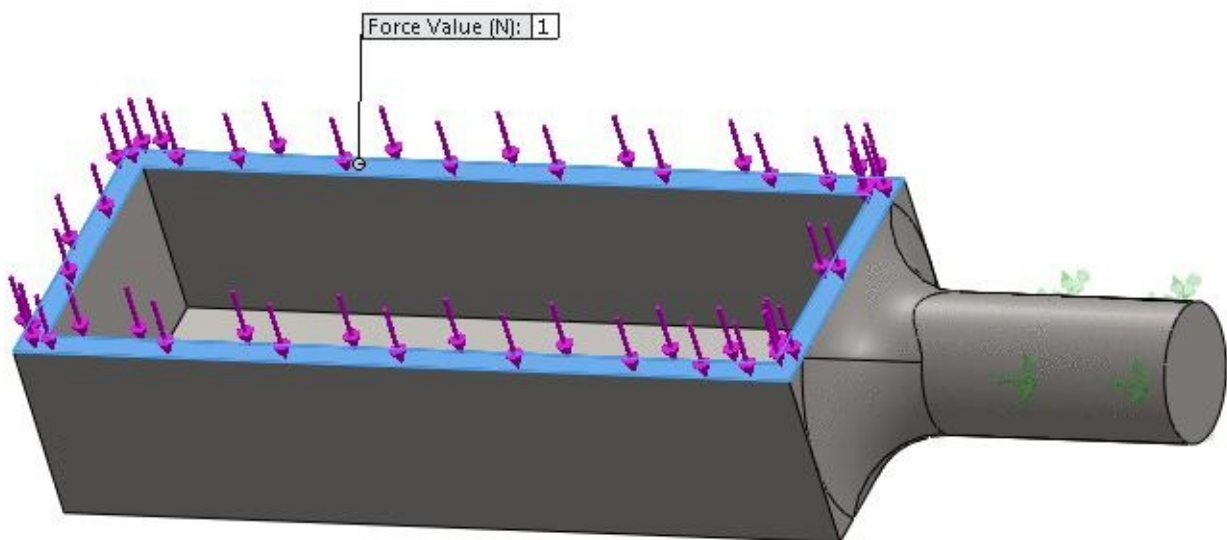


Figure-9. Face selected for applying load

- Click in the **Force Value** box in the **PropertyManager** or in the modeling area and enter the value as **980** which is equal to **100** Kilograms of weight. Refer to [Figure-10](#).

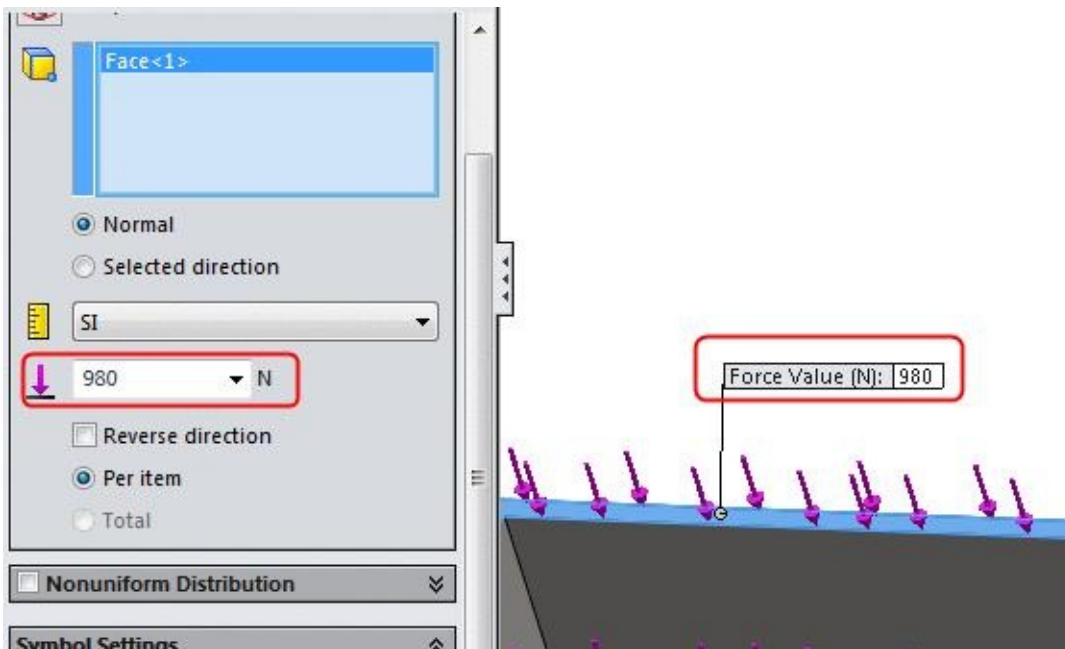


Figure-10. Value specified for load

- Click on the **OK** button from the **Force/Torque PropertyManager**. The force will be applied on the selected face.

For this particular problem, we do not require any connection to be applied since this is not an assembly. Now, we are ready to divide the model into small elements with the help of meshing.

Meshing the model

Note that you can skip this step if you do not want to explicitly define the element size for mesh. While running the analysis, system will automatically create the suitable mesh for you. To explicitly define the size of mesh elements, follow the steps given next.

- Click on the down arrow below the **Run** button in the **Ribbon**. The list of tools will be displayed; refer to [Figure-11](#).

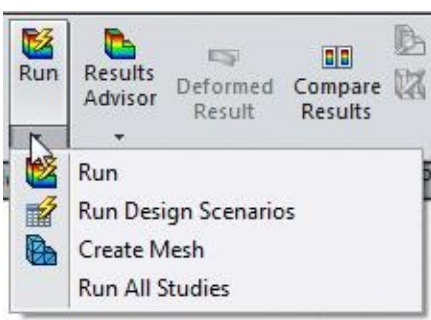


Figure-11. List of tools with Create Mesh tool

- Click on the **Create Mesh** tool from the list. The **Mesh PropertyManager** will be displayed; refer to [Figure-12](#).

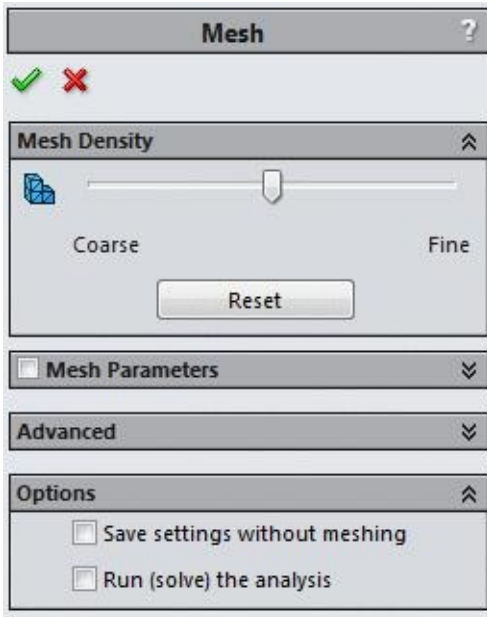


Figure-12. Mesh PropertyManager

- Click on the **Mesh Parameters** check box. The rollout will expand and the options related to element size and tolerance will be displayed; refer to [Figure-13](#).

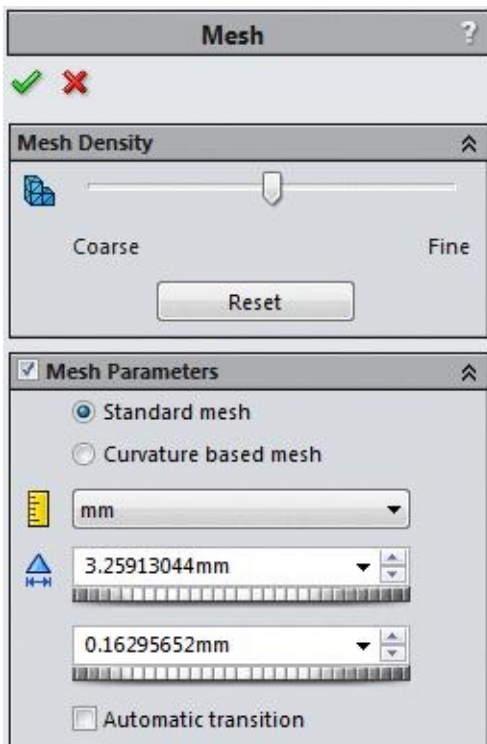


Figure-13. Mesh Parameters

- Set the desired values in the parameter boxes in the rollout and click on the **OK** button from the **PropertyManager**. Note that the smaller value given in the boxes will make the analysis slower because of more calculations. The meshed model will be displayed as in [Figure-14](#).

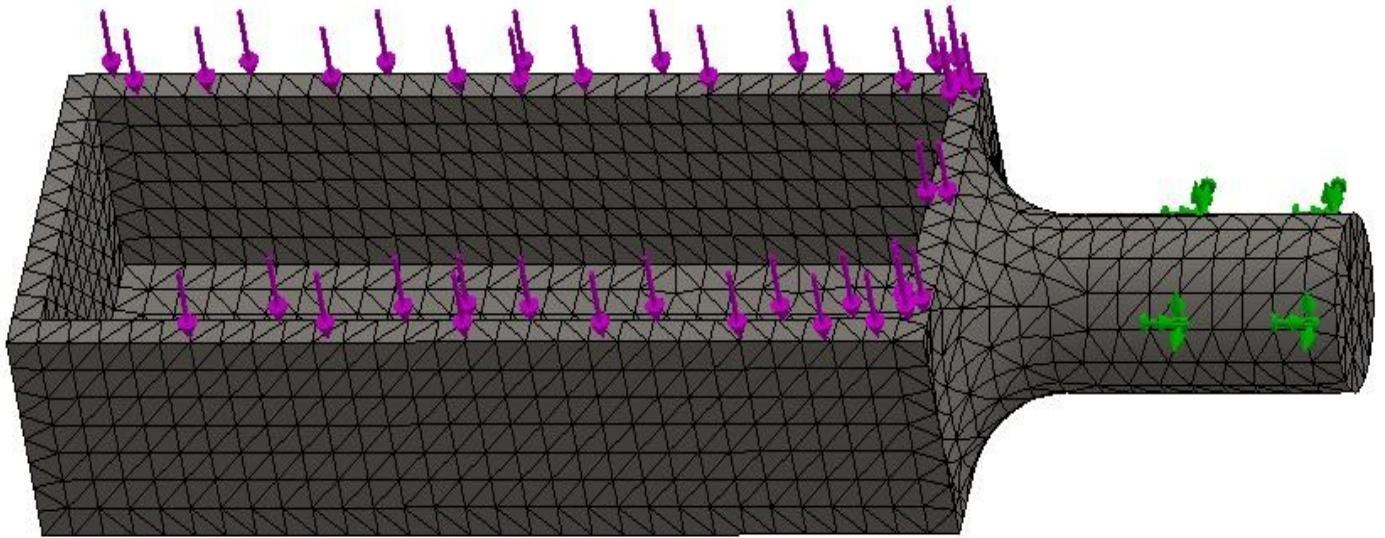


Figure-14. Meshed model

- To change the meshing for a particular area of the model, right-click on the **Mesh** node in the **Analysis Manager** displayed at the left of the modeling area. A shortcut menu will be displayed; refer to [Figure-15](#).

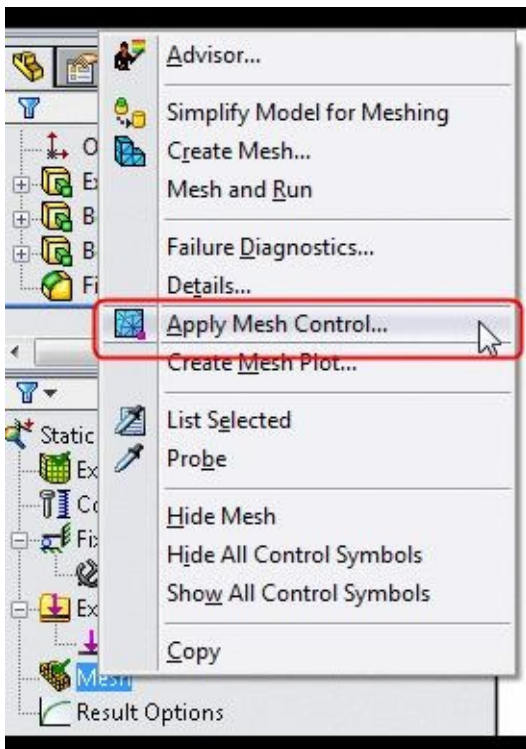


Figure-15. Meshing simplification

- Click on the **Apply Mesh Control** button. The **Mesh Control PropertyManager** will be displayed as shown in [Figure-16](#).

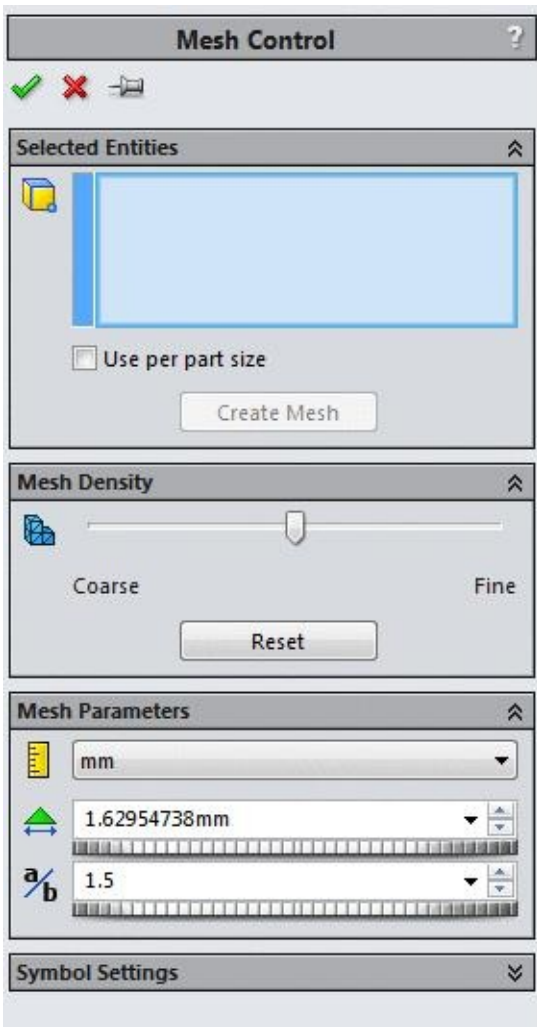


Figure-16. Mesh Control PropertyManager

- Select the faces of the model for which you want to change the mesh elements sizes; refer to [Figure-17](#).

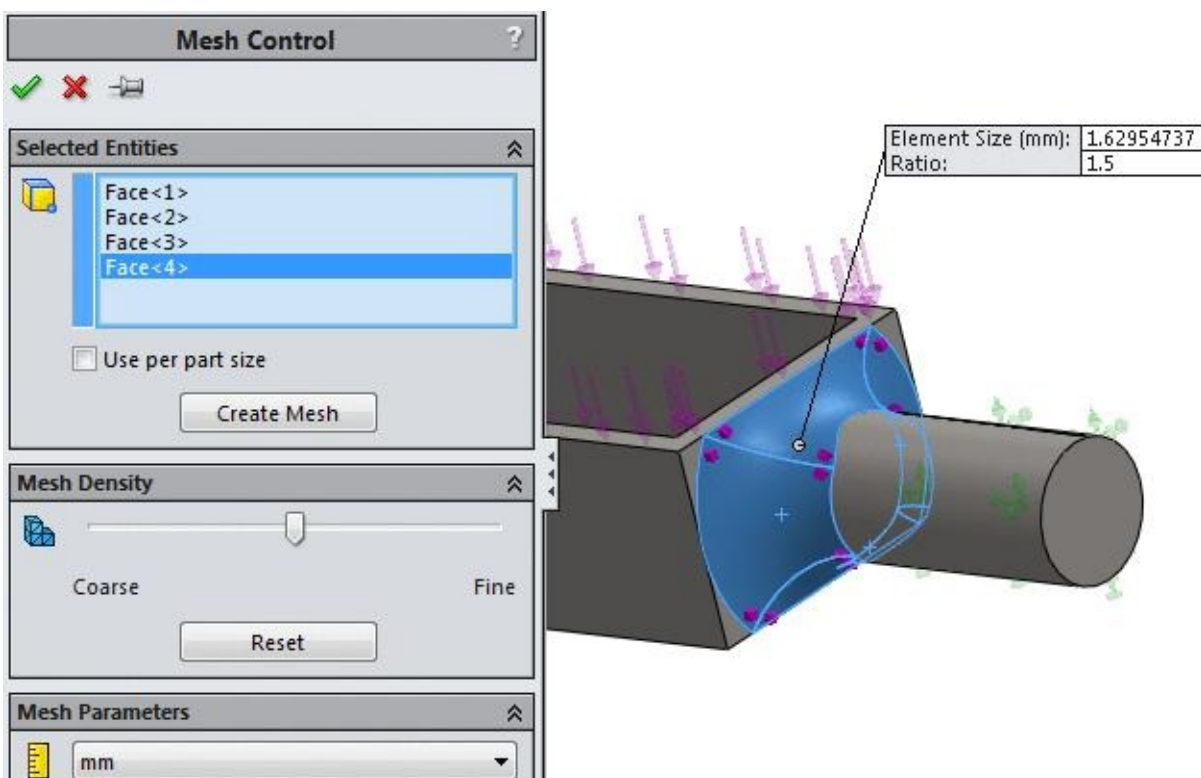


Figure-17. Faces for changing mesh size

- Move the slider in the **Mesh Density** rollout towards coarse or fine as per your requirement.
- Click on the **OK** button from the **PropertyManager** to create the updated mesh. Note that increasing or decreasing the element size will also affect the results. Decreasing the element size gives more accurate results.

Running the Analysis

- Click on the **Run** button from the **Ribbon**. The system will start solving the analysis.
- After the processing is complete, the result of analysis will be displayed; refer to [Figure-18](#).

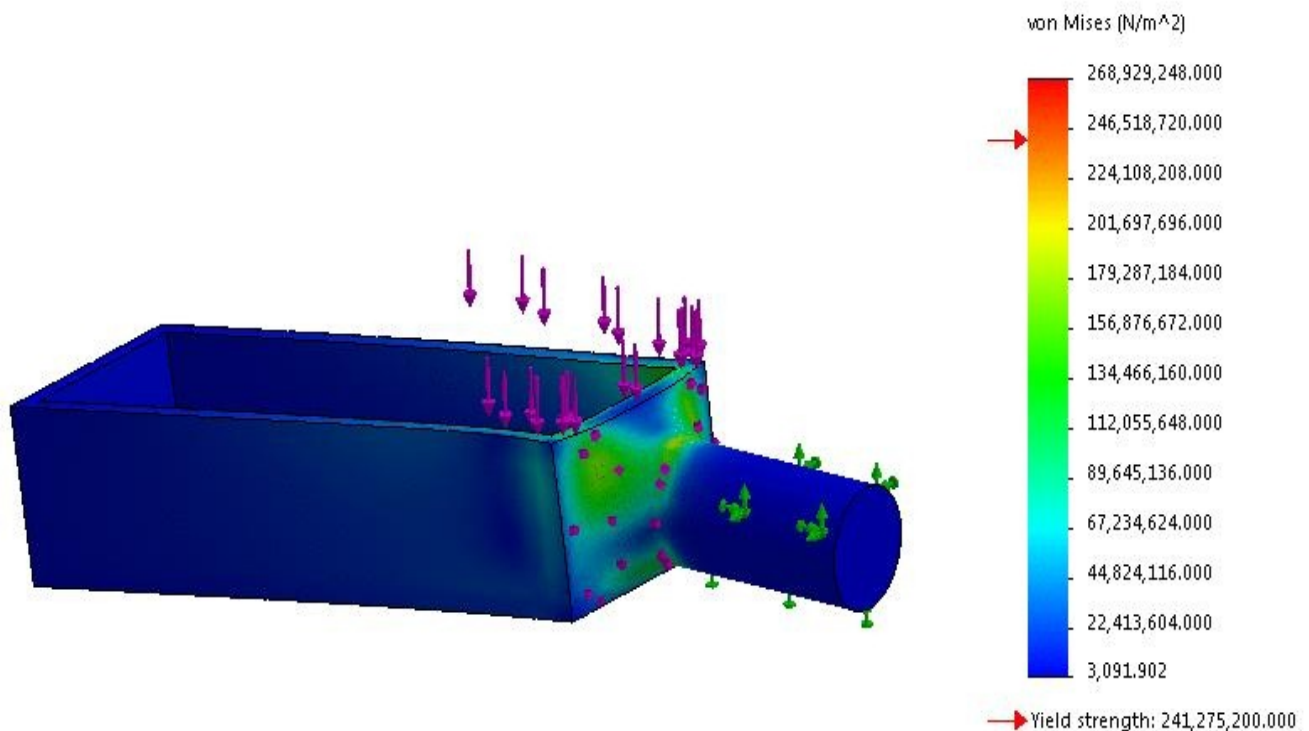


Figure-18. Analysis result

- By default, results related to stress (vonMises) are displayed in the modeling area.
- To check **Displacement** and **Strain** related results, double click on the respective option from the **Analysis Manager** in the left.

Results

- To check results other the default displayed, right-click on the **Results** node from the **Analysis Manager**. A shortcut menu will be displayed; refer to [Figure-19](#).

- Click on the **Define Factor Of Safety Plot** option to display the results related to **Factor Of Safety**. The **Factor of Safety PropertyManager** will be displayed as shown in [Figure-20](#).

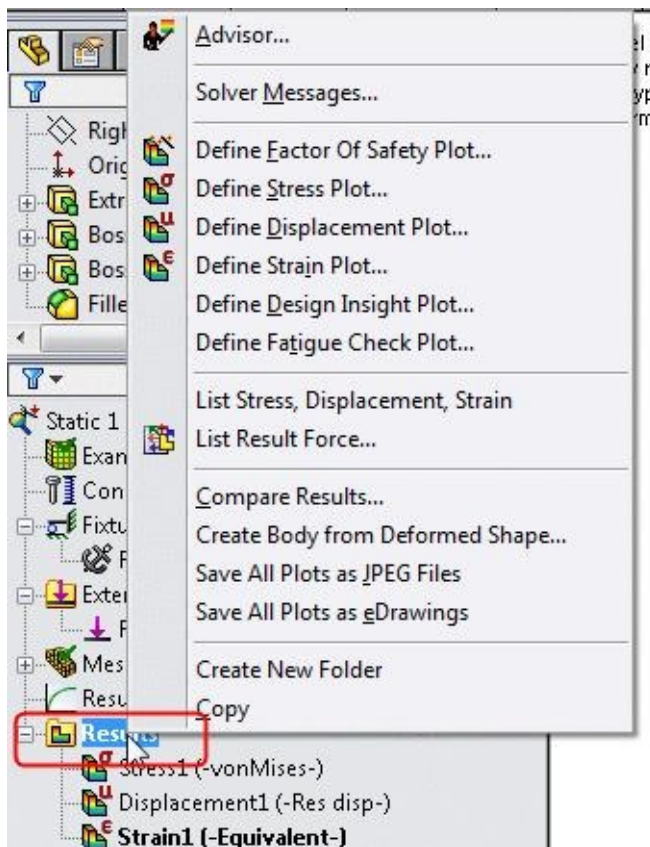


Figure-19. Shortcut menu for results

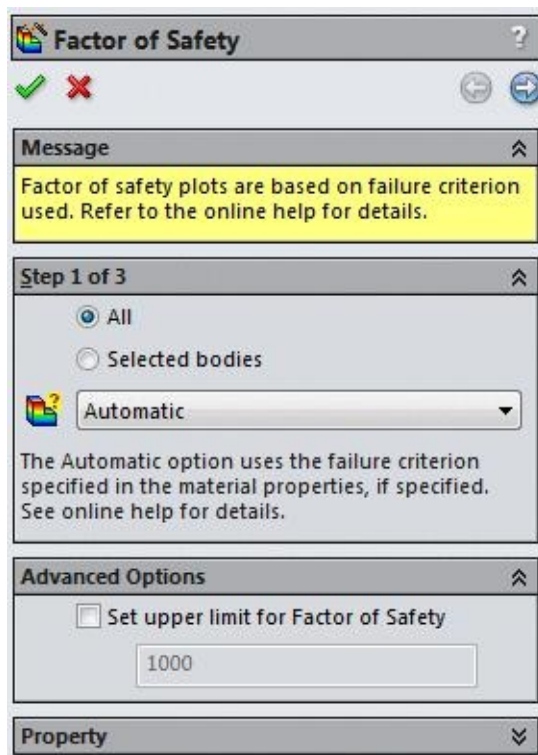


Figure-20. Factor of Safety PropertyManager

- If you have a special criteria for checking safety factor then select the respective option from the **Criterion** drop-down in the Step 1 of 3 rollout otherwise system will check the material properties and automatically decide the best criteria.
- Click on the **OK** button from the **PropertyManager**. The **Factor of Safety plot** will be displayed in the modeling area; refer to [Figure-21](#).

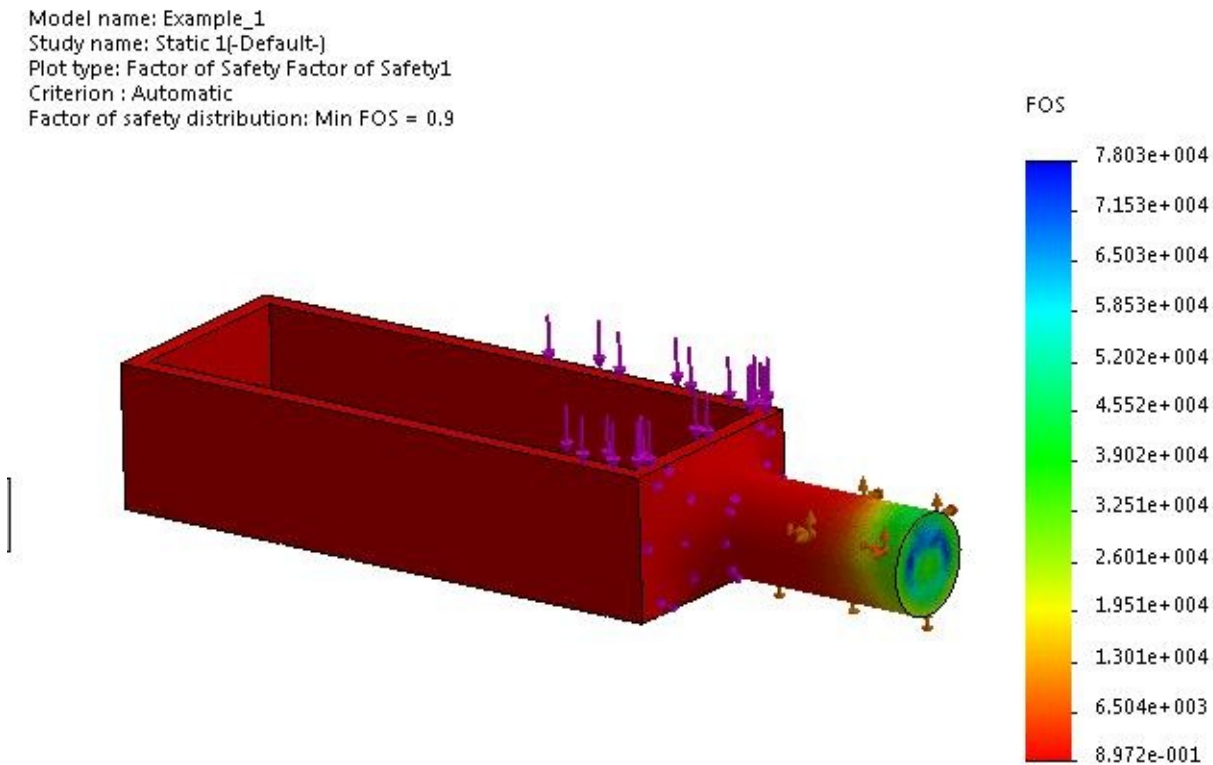


Figure-21. Factor of Safety plot

Optimizing product

- If you check in the results displayed above the model, the **Factor of safety distribution Min FOS = 0.9** is displayed which shows that the model is not safe at the locations displayed in dark red color. Note that for object to be safe for loading, **the FOS should be above 1**.
- Here, we need to either change the material or increase the thickness of the object to sustain the applied load.
- In this case, we change the material to **Alloy Steel**. To do so, right-click on the **Material** node in the **Analysis Manager**. A shortcut menu will be displayed as shown in [Figure-22](#).
- Select the **Apply/Edit Material** from the menu. The **Material** dialog box will be displayed.
- Select the **Alloy Steel** material from the left table and click on the **Apply** button to apply the material.
- Click on the **Close** button from the dialog box to close it.

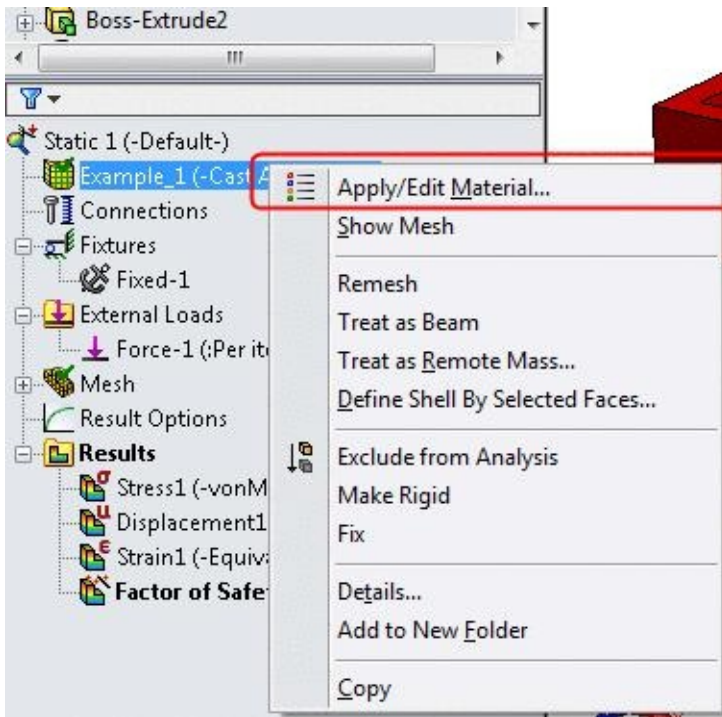


Figure-22. Shortcut menu for material

- Since the material is change, we need to run the analysis again. To do so, click on the **Run** button from the **Ribbon**. The analysis will re-run and results will be displayed.
- Double-click on the **Factor of Safety1 (-FOS-)** node from the **Analysis Manager**. The results will be displayed as shown in [Figure-23](#).

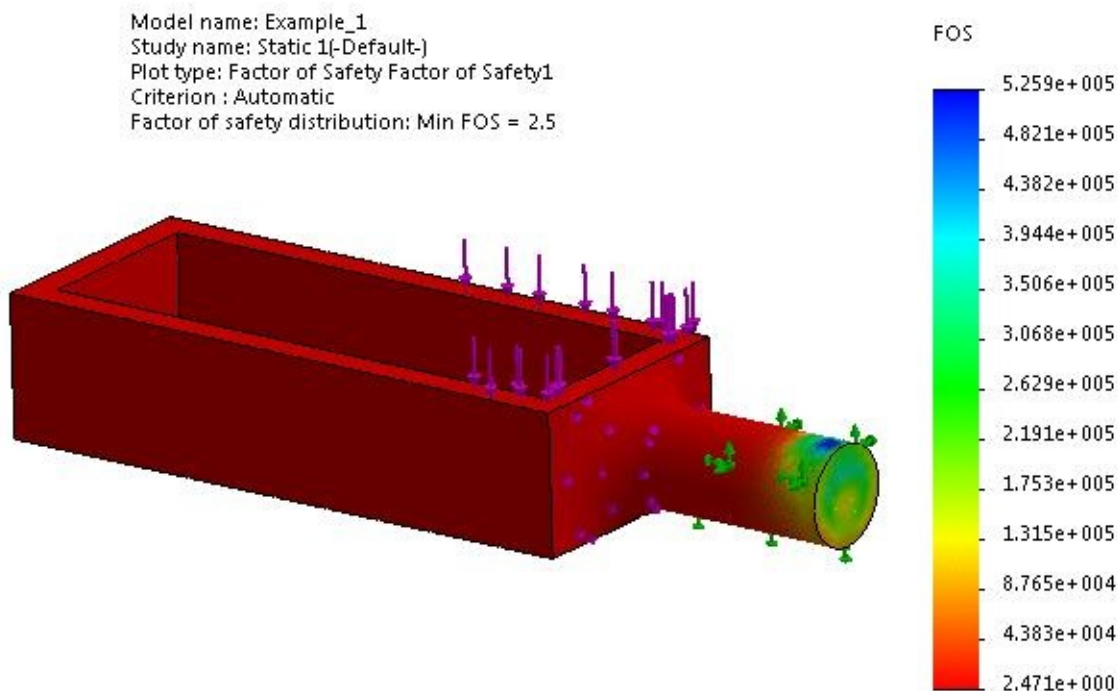


Figure-23. Updated Factor of Safety results

- Now, the factor of safety is **2.5** i.e. above **1**. So, our product can sustain the applied load.

SHELL MANAGER

In SolidWorks Simulation 2015, you can also manage all the shells at a single place by using the Shell Manager. Shells are generally used to represent parts that have very low thickness like sheetmetal parts. The advantage of using shells lies in element type used for them during meshing. For solid objects 3D elements are used but for shells 2D elements like linear triangular and parabolic triangular elements are used. The analysis performed using 2D elements is faster. The procedure to use the **Shell Manager** is given next.

- Before we start using **Shell Manager**, we have a surface model as shown in [Figure-24](#). This model is a representation of sheetmetal part with uniform thickness of 5 mm.

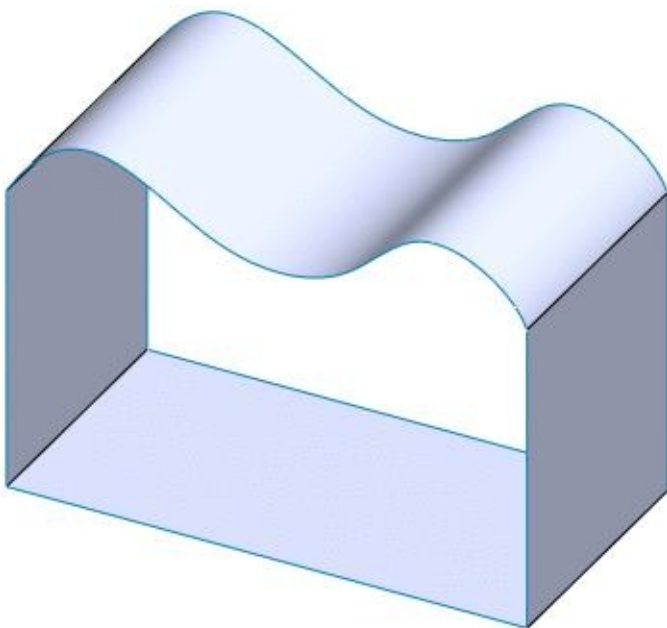


Figure-24. Surface model

- To assign thickness to the model, click on the **Shell Manager** tool from the **Simulation** tab of **Ribbon**. The Shell Manager will be displayed with Shell Manager pane; refer to [Figure-25](#).

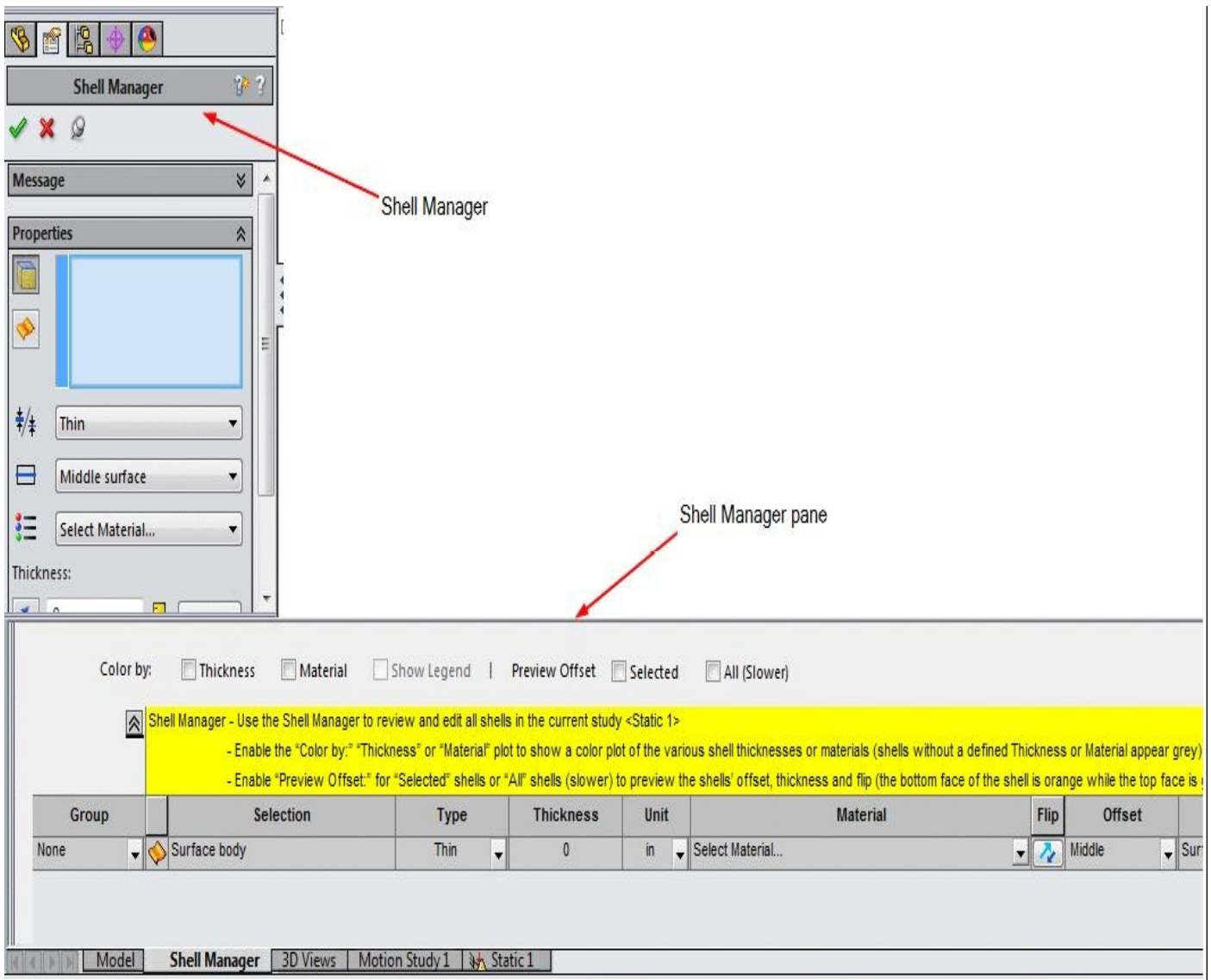


Figure-25. Shell Manager

- If the object has uniform thickness then click in the cell under **Thickness** column in the **Shell Manager** pane and enter the desired thickness value; refer to [Figure-26](#). Make sure that you have selected right unit from the **Unit** column in the pane.

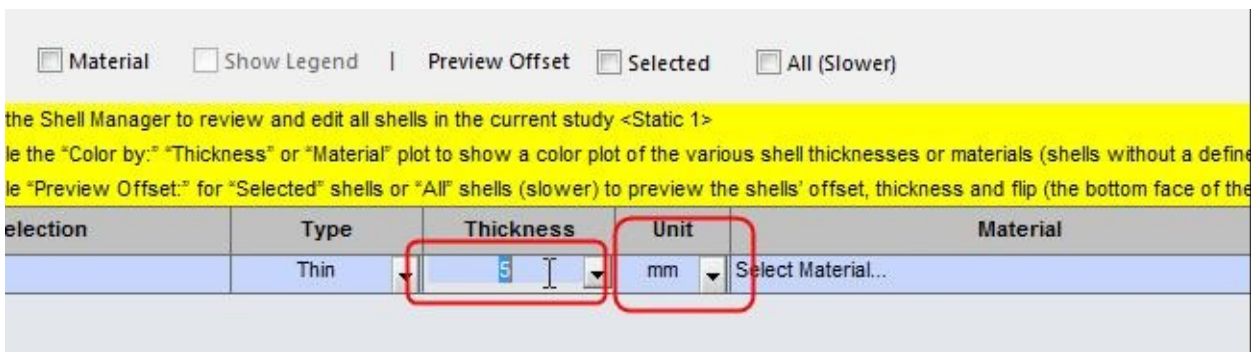


Figure-26. Uniform thickness applied

- If you want to specify different thickness to any of the face then click in the selection box in **Properties** rollout in the **Shell Manager** and select the exclusive faces; refer to

Figure-27.

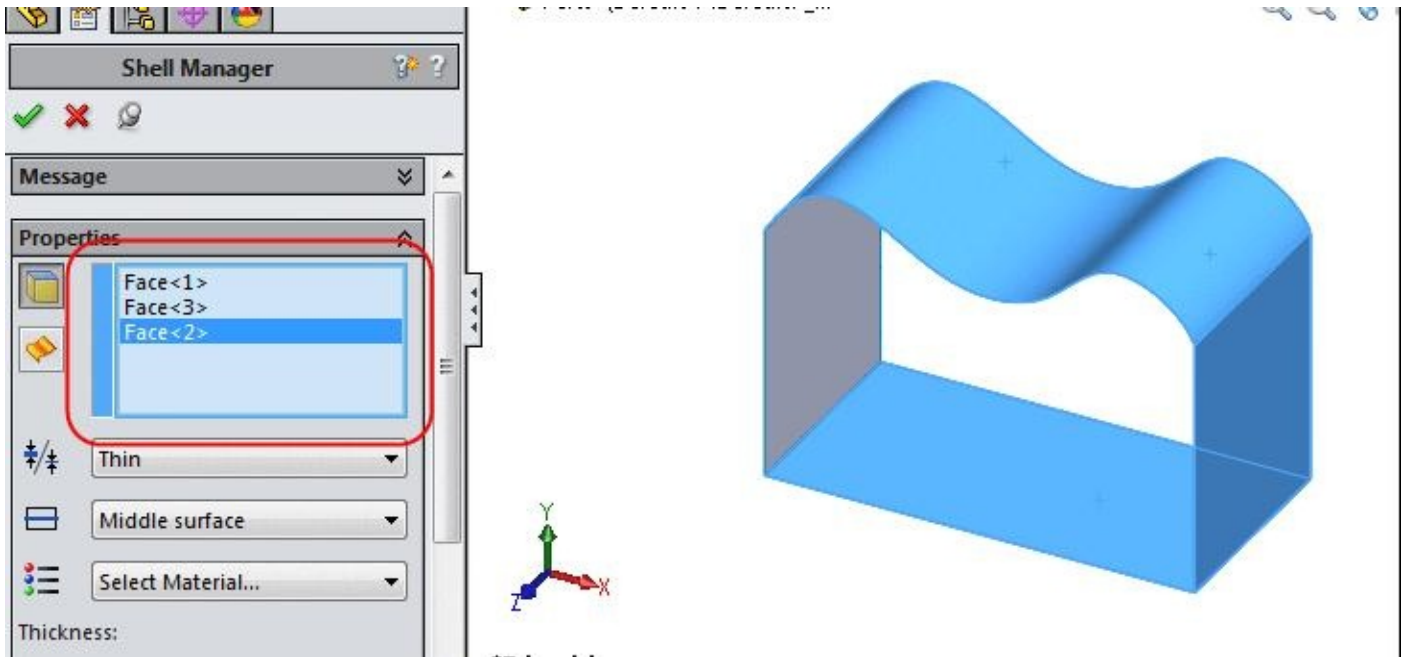


Figure-27. Selection box in Properties rollout

- Click on the **OK** button from the **Shell Manager**. The surfaces will be displayed in the **Shell Manager** pane; refer to [Figure-28](#).

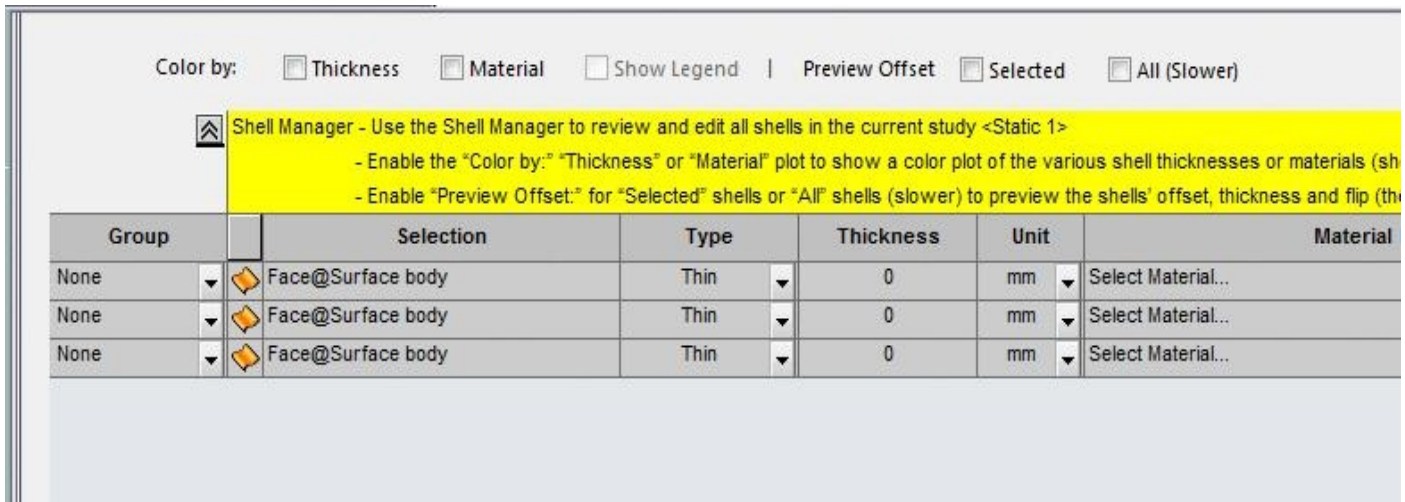


Figure-28. Shell Manager pane

- One by one, enter the desired sheet thickness for each face. Make sure that you press **ENTER** after specifying thickness in the cells.
- Similarly, you can select other parameters like material, offset type etc. for individual faces. [Figure-29](#) shows the faces assigned with different materials.

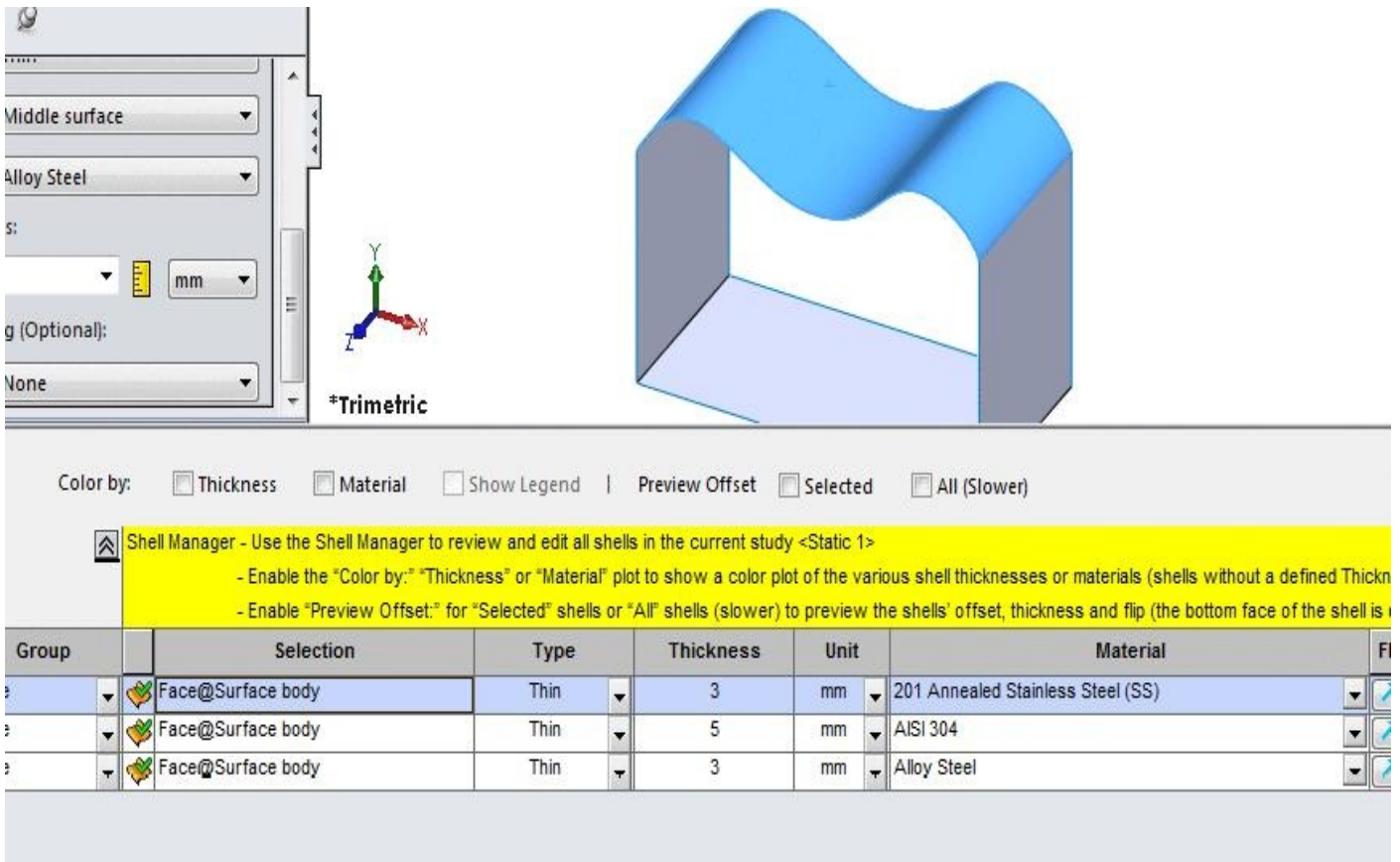


Figure-29. Faces with different material and thickness

- After specifying the parameters, click on the **OK** button from the **Shell Manager**. Now, you can continue with the analysis just like a solid part (with hybrid material).

PERFORMING STATIC ANALYSIS USING STUDY ADVISOR

Earlier in this chapter, we have performed the static analysis by selecting each of the required option from the list of tools in the **Ribbon**. We can use the **Study Advisor** to perform all the steps related to analysis. The steps to use the Study Advisor are given next.

Starting the Static Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **Study Advisor** tool from the drop-down. The **Simulation Advisor** will be displayed in the right of the application window; refer to [Figure-30](#).
- Click on the **Next** link button from the **Simulation Advisor**. The study page of the

Simulation Advisor will be displayed as shown in [Figure-31](#).

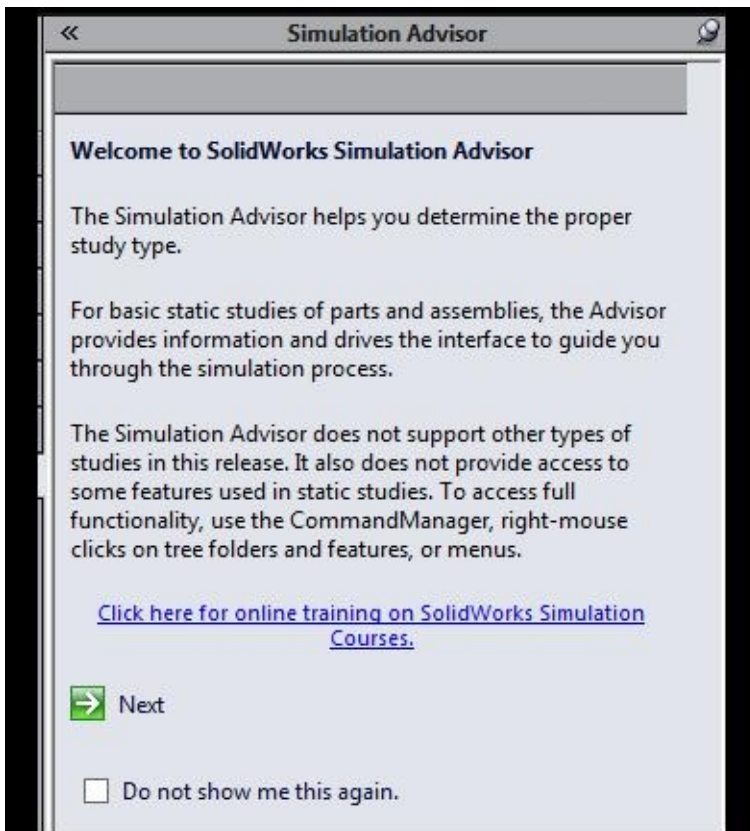


Figure-30. Simulation Advisor

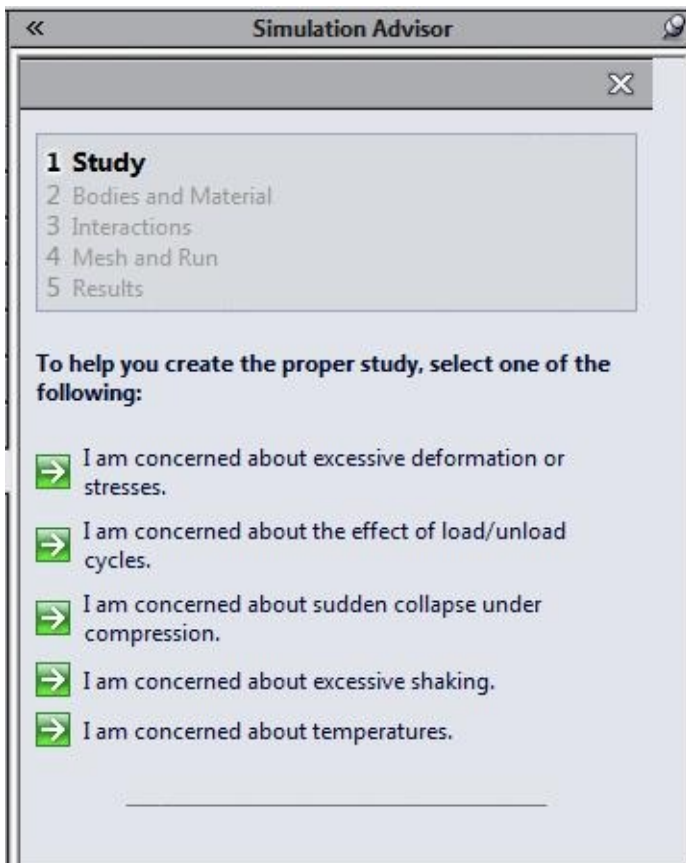


Figure-31. Study page of Simulation Advisor

- Currently, we are planning to perform the stress analysis. For that, click on the **I am concerned about excessive deformation or stresses link** button. The **second page of Study** will be displayed in the **Simulation Advisor**; refer to [Figure-32](#).

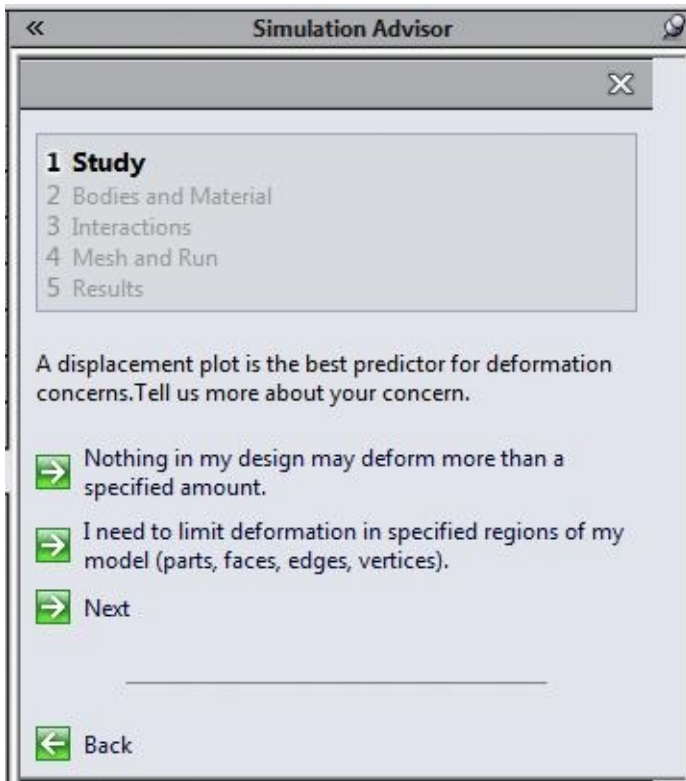


Figure-32. Second page Study

- Using the options in this page, we can check the deformation plot for the present analysis. Deformation plot displays the visualization of deformation for the analysis.

1. If you want to check the deformation for a specific part/face/edge/vertex then click on the **I need to limit deformation in specified regions of my model(parts, faces, edges, vertices)** link button. The **Sensor PropertyManager** will be displayed as shown in [Figure-33](#).

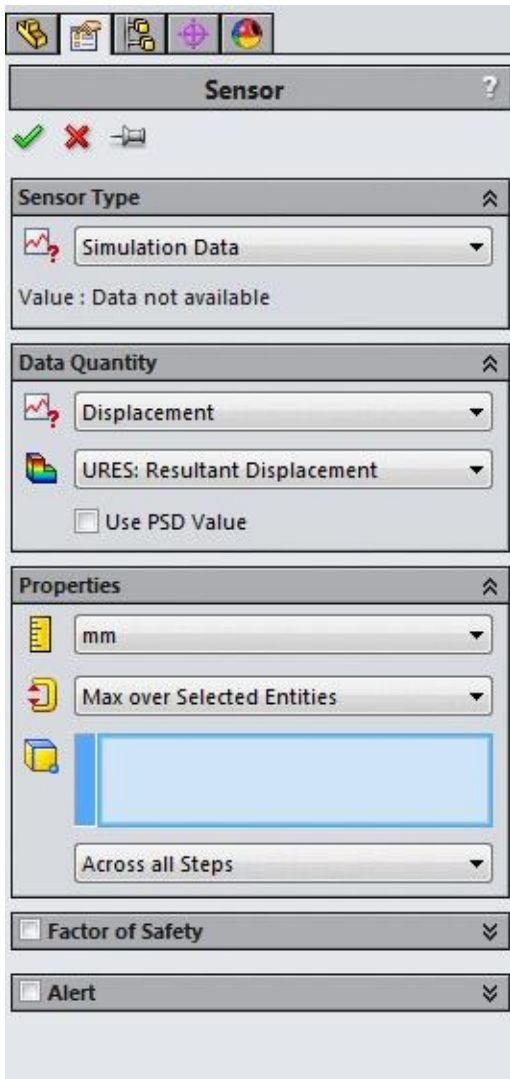


Figure-33. Sensor PropertyManager

2. Select the part/face/edge/vertex for which you want to check the deformation; refer to [Figure-34](#) and click on the **OK** button from the **PropertyManager**. The page related to material failure will be displayed in the **Simulation Advisor**.

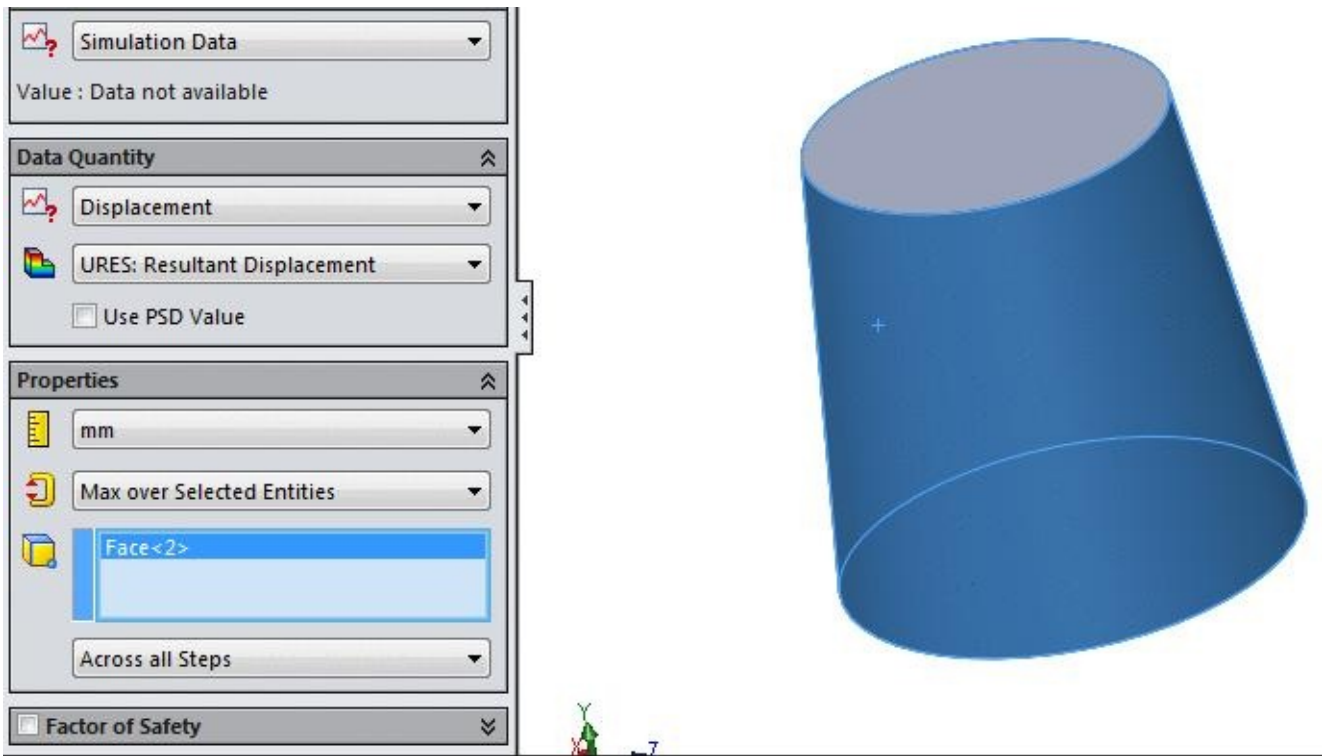


Figure-34. Face selected for deformation testing

- If you want to specify limit for deformation then click on the **Nothing in my design may deform more than a specified amount**. The **Sensor PropertyManager** will be displayed.

1. Click on the **Alert** rollout to expand it. The options in the rollout will be displayed as shown in [Figure-35](#).

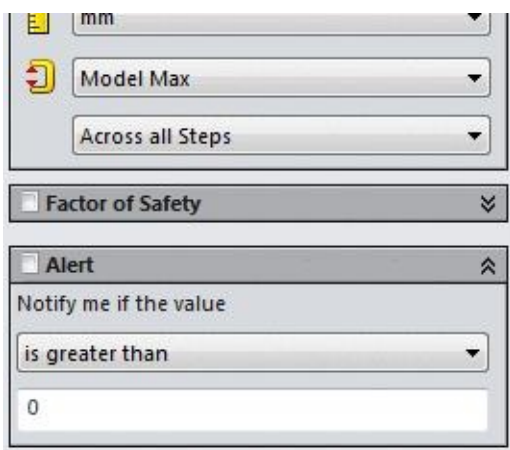


Figure-35. Sensor PropertyManager with Alert options

2. Click in the edit box to specify the limit of deformation. If the deformation crosses the specified value then an alert will be displayed.
3. Click **OK** button from the **PropertyManager**. Page related to material failure will be displayed in the **Simulation Advisor**; refer to [Figure-36](#).

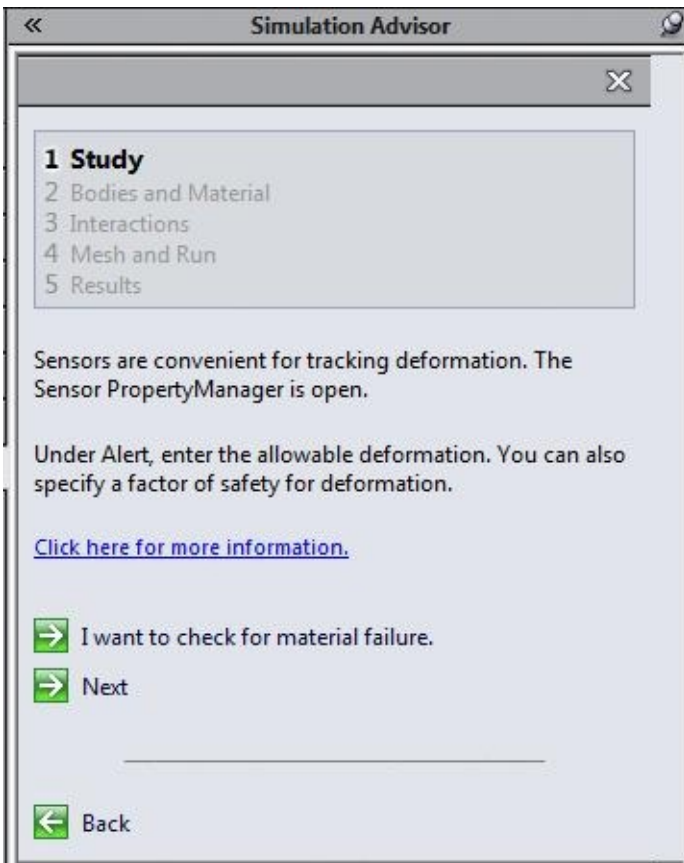


Figure-36. Simulation Advisor with page related to material failure

- Click on the **I want to check material failure link** button to check for material failure. The next page for material failure will be displayed in the **Simulation Advisor**; refer to [Figure-37](#).

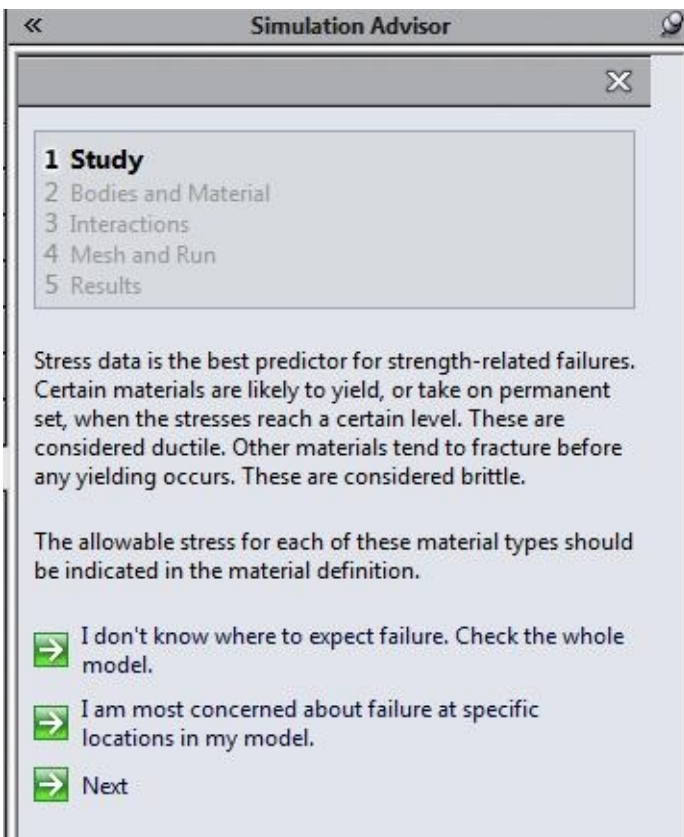


Figure-37. Simulation Advisor with another page related to material failure

- Click on the **I don't know where to expect failure. Check the whole model** link button to check the whole model for material failure. You can select the **I am most concerned about failure at specific locations in my model** link button if you want to check the material failure for a specific portion. In this case, we have selected the first link button. On doing so, again the **Sensor PropertyManager** is displayed where you can specify the alert for maximum stress. Note that the maximum bearable stress is generally obtained by material properties.
- Click on the **OK** button from the **PropertyManager** and then click on the **Next** link button from the **Simulation Advisor**, the name of the study that suits your requirements will be displayed; refer to [Figure-38](#).

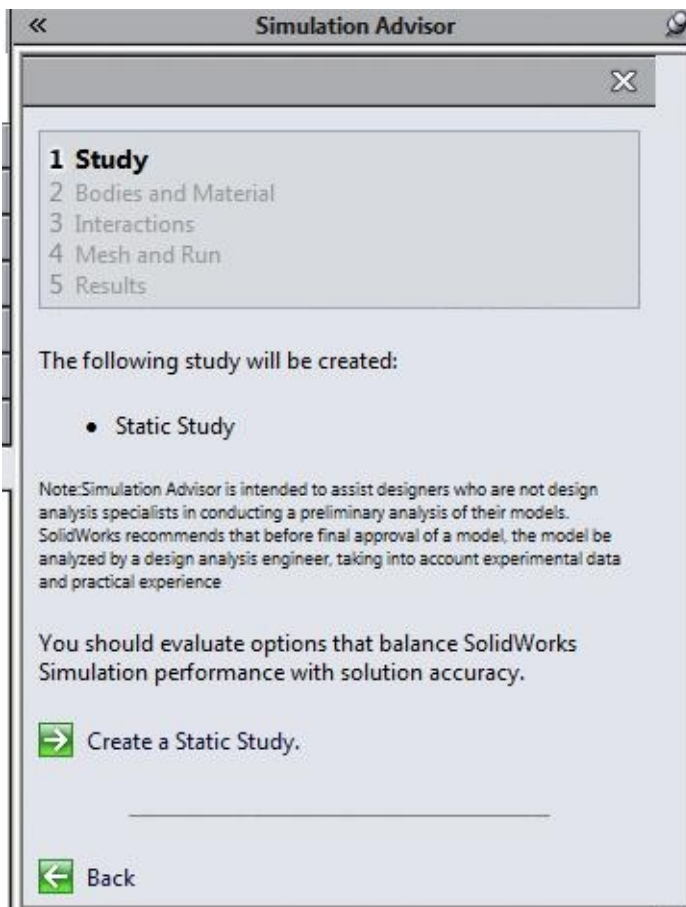


Figure-38. Simulation Advisor with Study name

- Click on the **Create a Static Study** link button and then click on the **Next** link button from the **Simulation Advisor**. The **Bodies and Material** page will be displayed in the **Simulation Advisor**.
- Click on the **Next** button from the pages related to symmetry will be displayed; refer to [Figure-39](#). Note that if you apply symmetry then calculation time for analysis will be reduced.

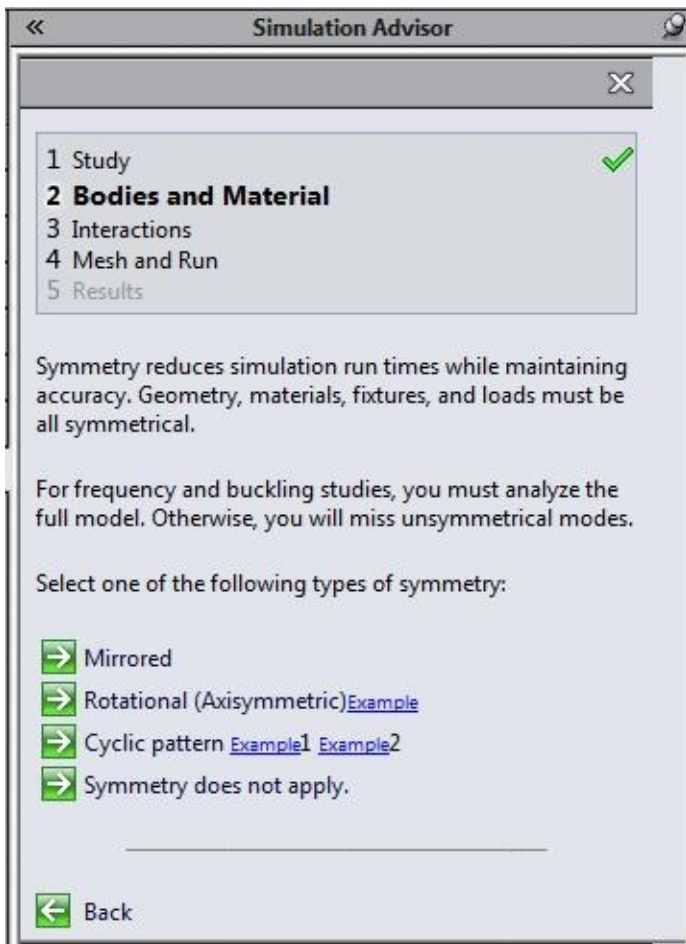


Figure-39. Simulation Advisor with symmetry page

- Click on the desired link from the **Simulation Advisor**. For the current example, click on the **Mirrored** link button and then click on the **Add symmetry fixture on solid faces** link button from the **Simulation Advisor**. The **Fixture PropertyManager** will be displayed; refer to [Figure-40](#).
- You can apply the symmetry option as discussed in previous chapter.
- After you are finished with the symmetry then click on the **Finished with Symmetry** link button. The page related to body setup will be displayed; refer to [Figure-41](#).

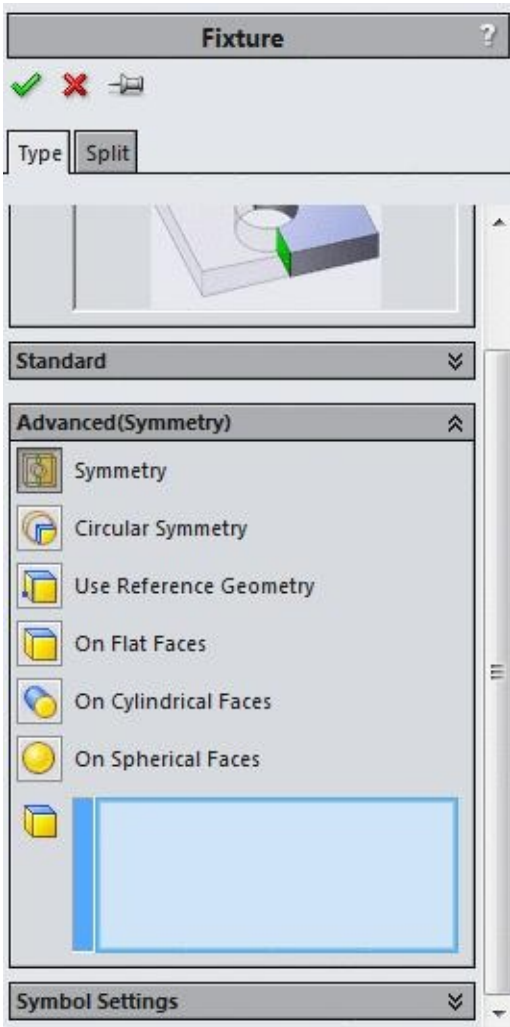


Figure-40. Fixture PropertyManager with symmetry option

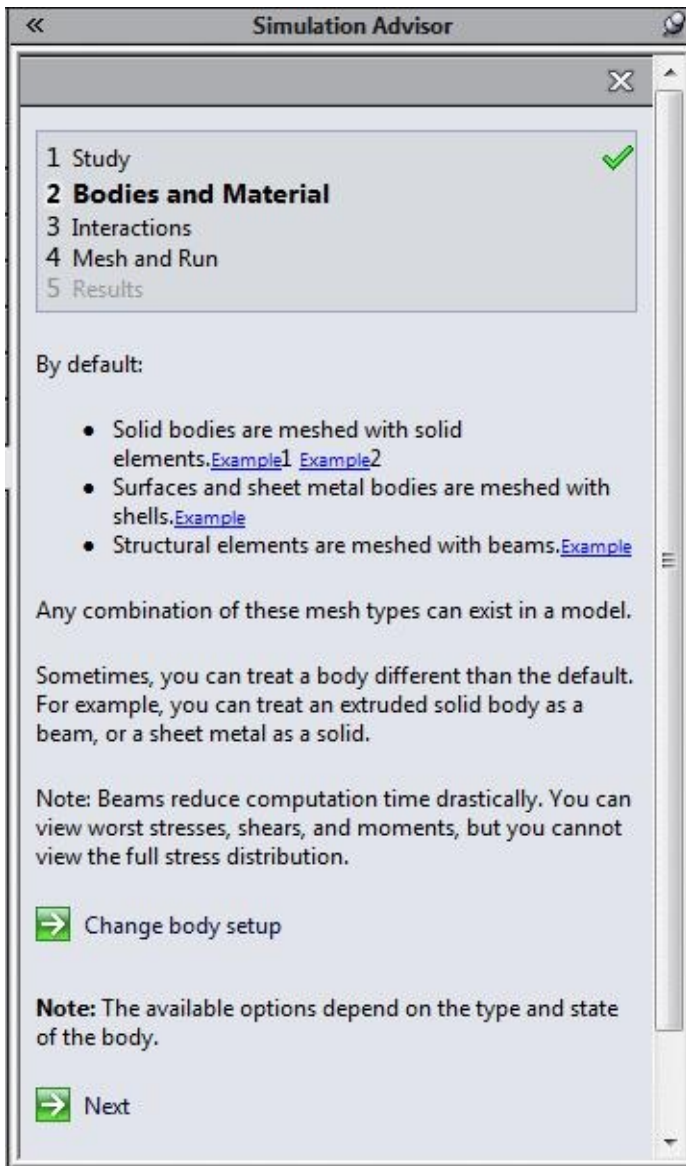


Figure-41. Page related to body setup

- Click on the **Change body setup** link button to specify setting for the body. You are displayed a message box saying then you need to right-click on the body and select the Treat as option. Right-click on the name of the model in the **Analysis Manager** and select the desired treat as option; refer to [Figure-42](#). The option related to the selected option will be displayed in the form of **PropertyManager**. Act accordingly.
- Now, click on the **Next** button from the body setup page of **Simulation Advisor**. The material page will be displayed; refer to [Figure-43](#).

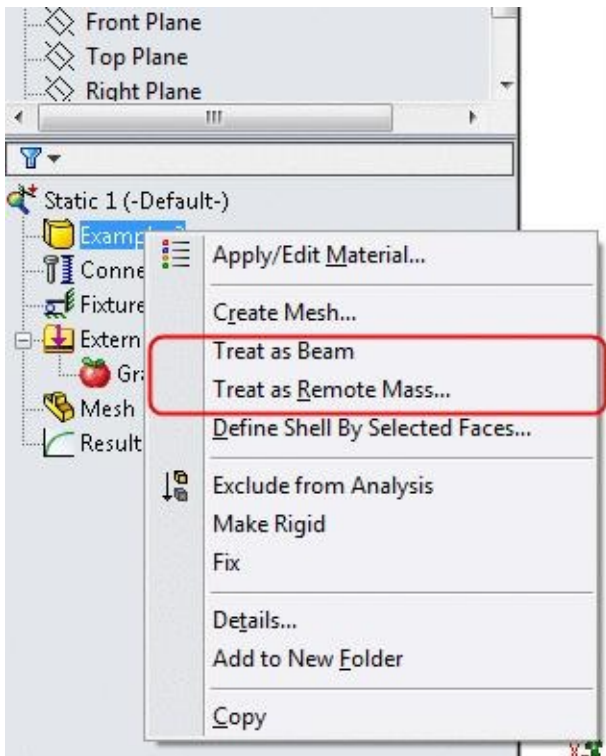


Figure-42. Treat as options

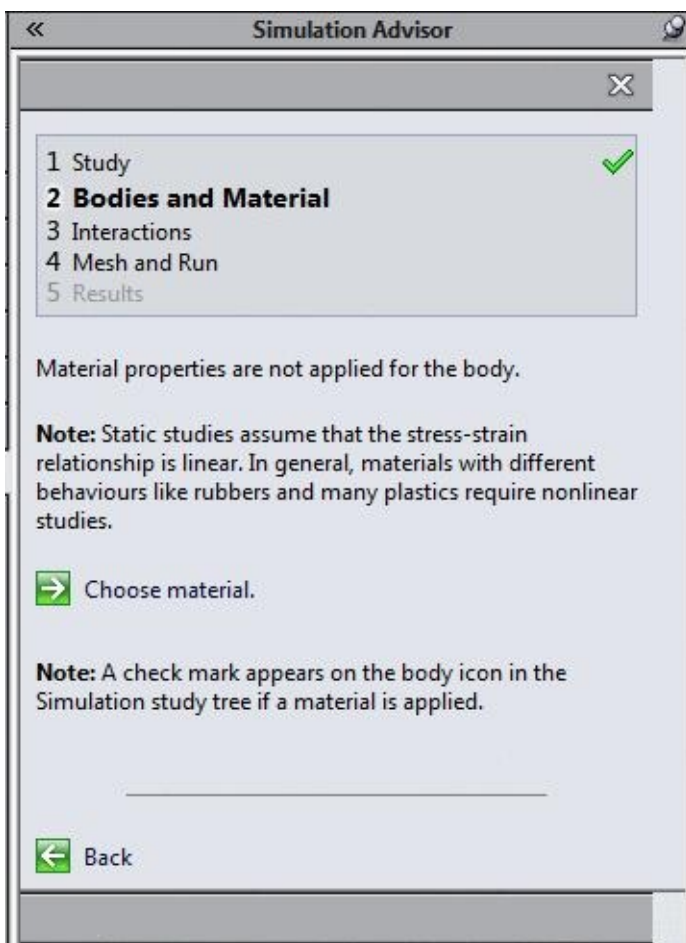


Figure-43. Material page of Simulation Advisor

- Click on the **Choose material** link button to specify the material for the current model.

The Material dialog box will be displayed as discussed earlier. Select the material, click on the **Apply** button and then click on the close button. The applied material will be displayed in the **Simulation Advisor**.

- Click on the **Next** button from the **Simulation Advisor**. The **Interactions** page will be displayed; refer to [Figure-44](#).

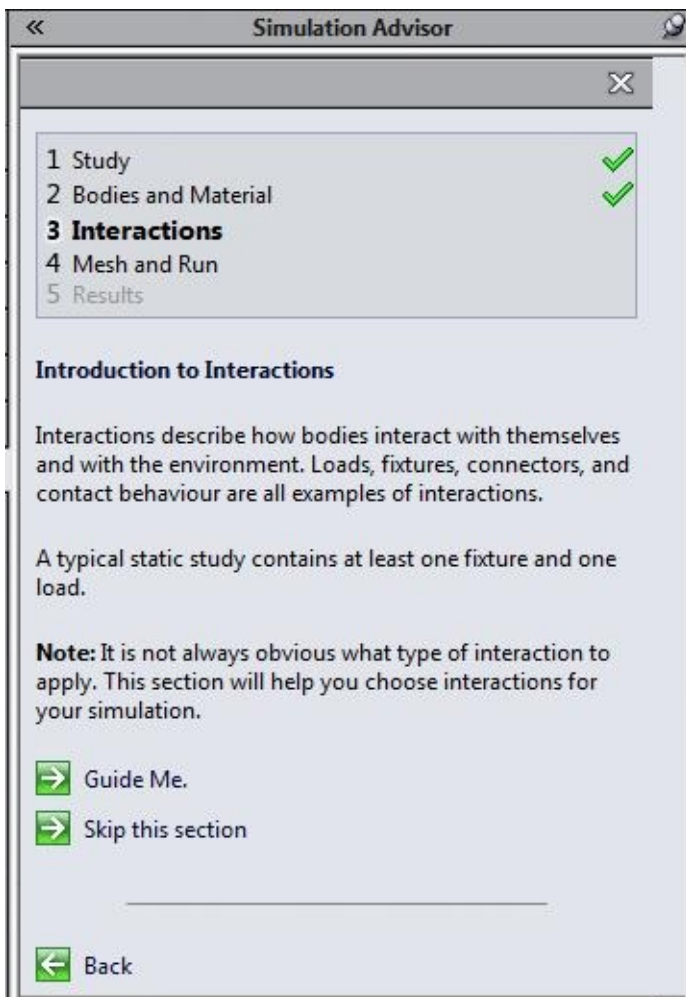


Figure-44. Simulation Advisor with Interaction page

Now onwards, the path of using **Simulation Advisor** becomes very confusing. So, we will go back to **Ribbon** and select the desired option from there.

- Click on the **Fixtures Advisor** button from the **Ribbon**. The options related to fixtures will be displayed in the **Simulation Advisor**; refer to [Figure-45](#).

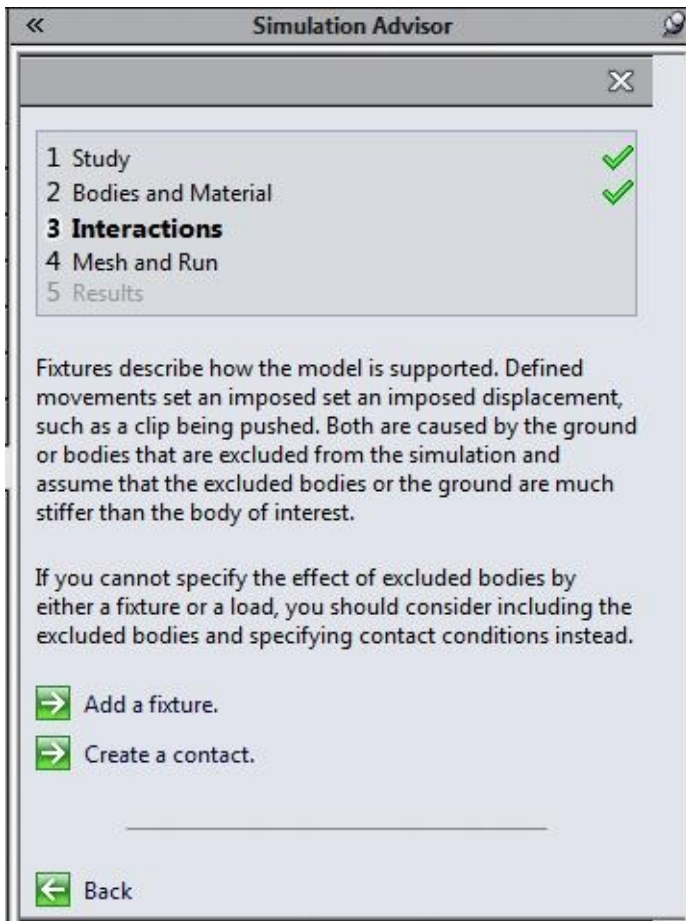


Figure-45. Options for fixture in Simulation Advisor

- Click on the **Add a fixture** link button. The **Fixture PropertyManager** will be displayed as discussed earlier.
- Select the bottom face of the current model; refer to [Figure-46](#). And click on the **OK** button from the **PropertyManager** to apply the fixture.

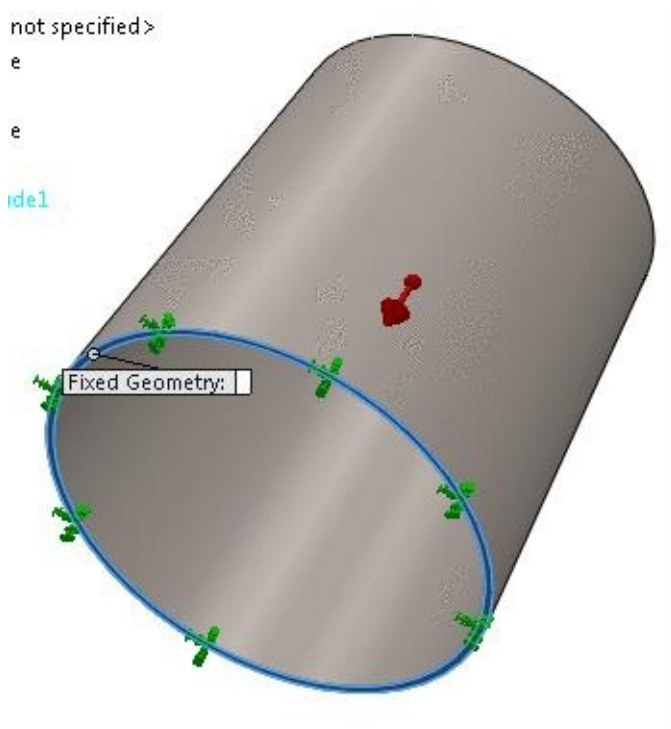


Figure-46. Face selected for fixture

- Click on the **External Loads Advisor** button from the **Ribbon**. The loads page of **Simulation Advisor** will be displayed; refer to [Figure-47](#).

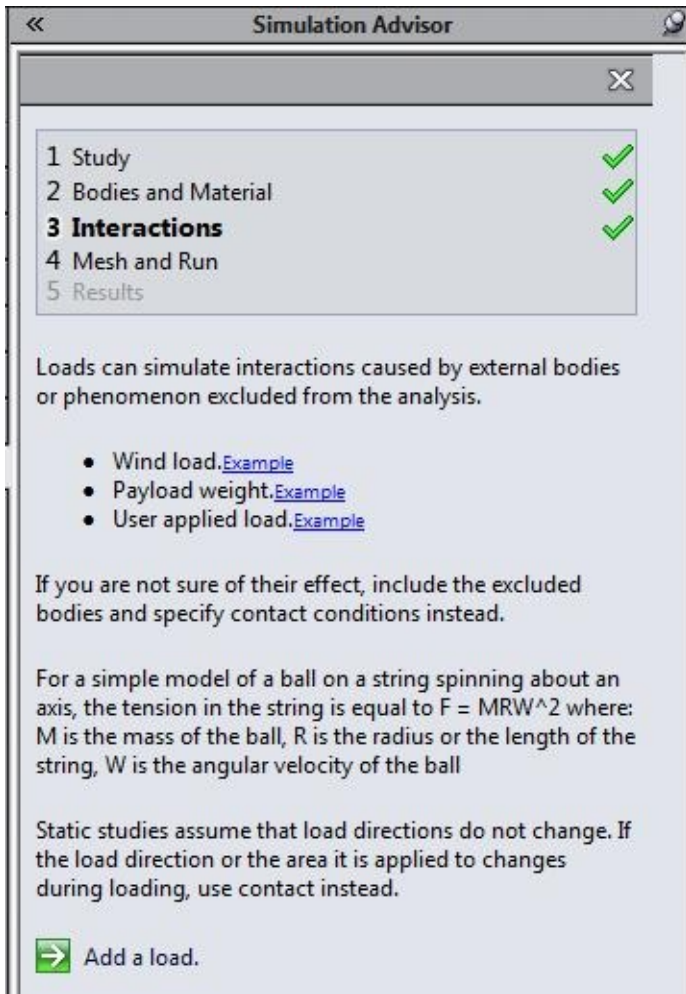


Figure-47. Loads page of Simulation Advisor

- Click on the **Add a load** link button. The **Force/Torque PropertyManager** will be displayed as discussed earlier.
- Select the top face of the model and specify the value of force as **392 N**; refer to [Figure-48](#).
- Click on the **OK** button from the **PropertyManager** to apply the force.

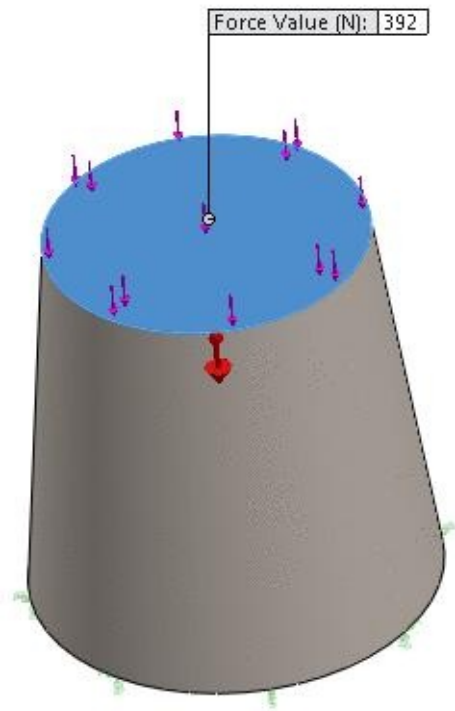


Figure-48. Force applied on top face

- Click on the **Done - Continue to next Advisor section** link button from the **Simulation Advisor**. The **Mesh and Run** page of **Simulation Advisor** will be displayed as shown in [Figure-49](#).

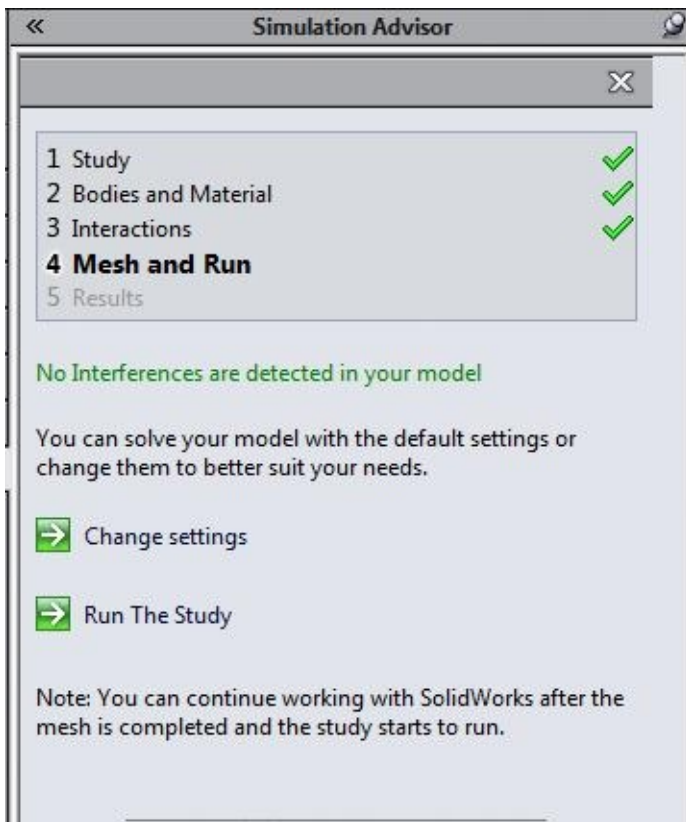


Figure-49. Mesh and Run page

- Click on the **Change settings** link button and then specify the desired mesh sizes for global or local areas of the model.
- In our case, we use the default settings and click on the **Run The Study** link button. First meshing will be processes and then the analysis will be performed. After the analysis is completed, then the results page will be displayed in the **Simulation Advisor** as shown in [Figure-50](#).

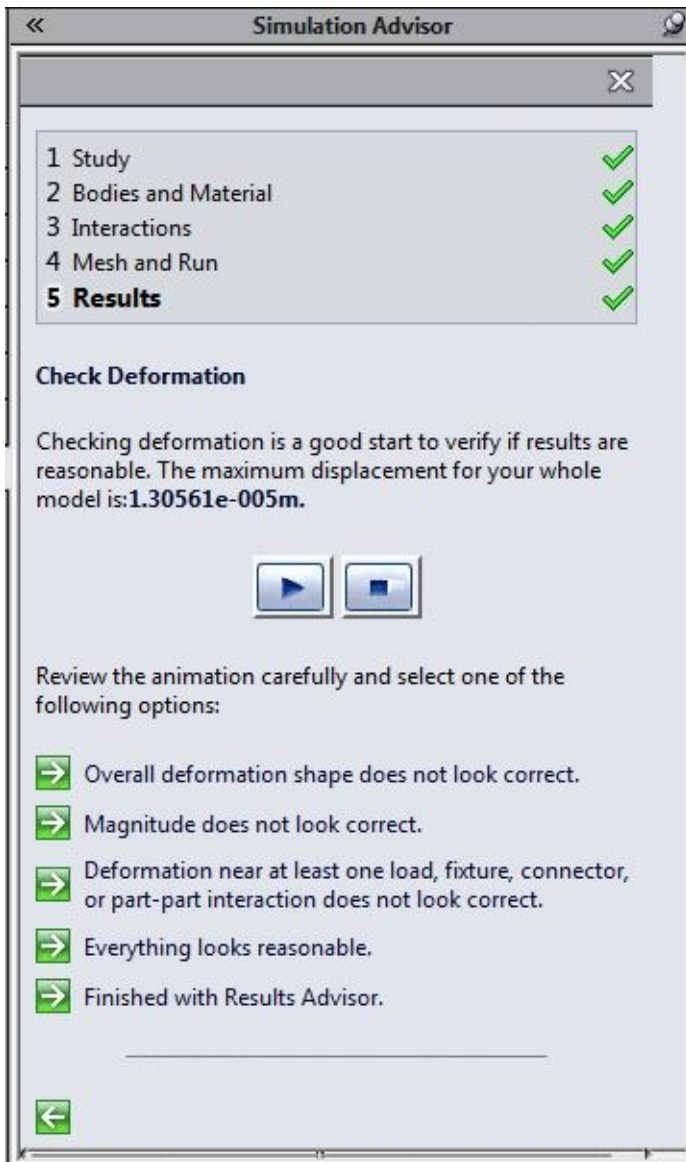


Figure-50. Result page of Simulation Advisor

- Click on the **Play** button from the **Simulation Advisor** to check the results. Click on the **Stop** button to stop animation.
- Click on the **Everything looks reasonable** link button to check the other results.
- Next, Click on the **Material breaking or yielding** link button and then **Show Factor of Safety plot** link button to plot the **Factor of Safety** result. The result will be displayed; refer to [Figure-51](#).

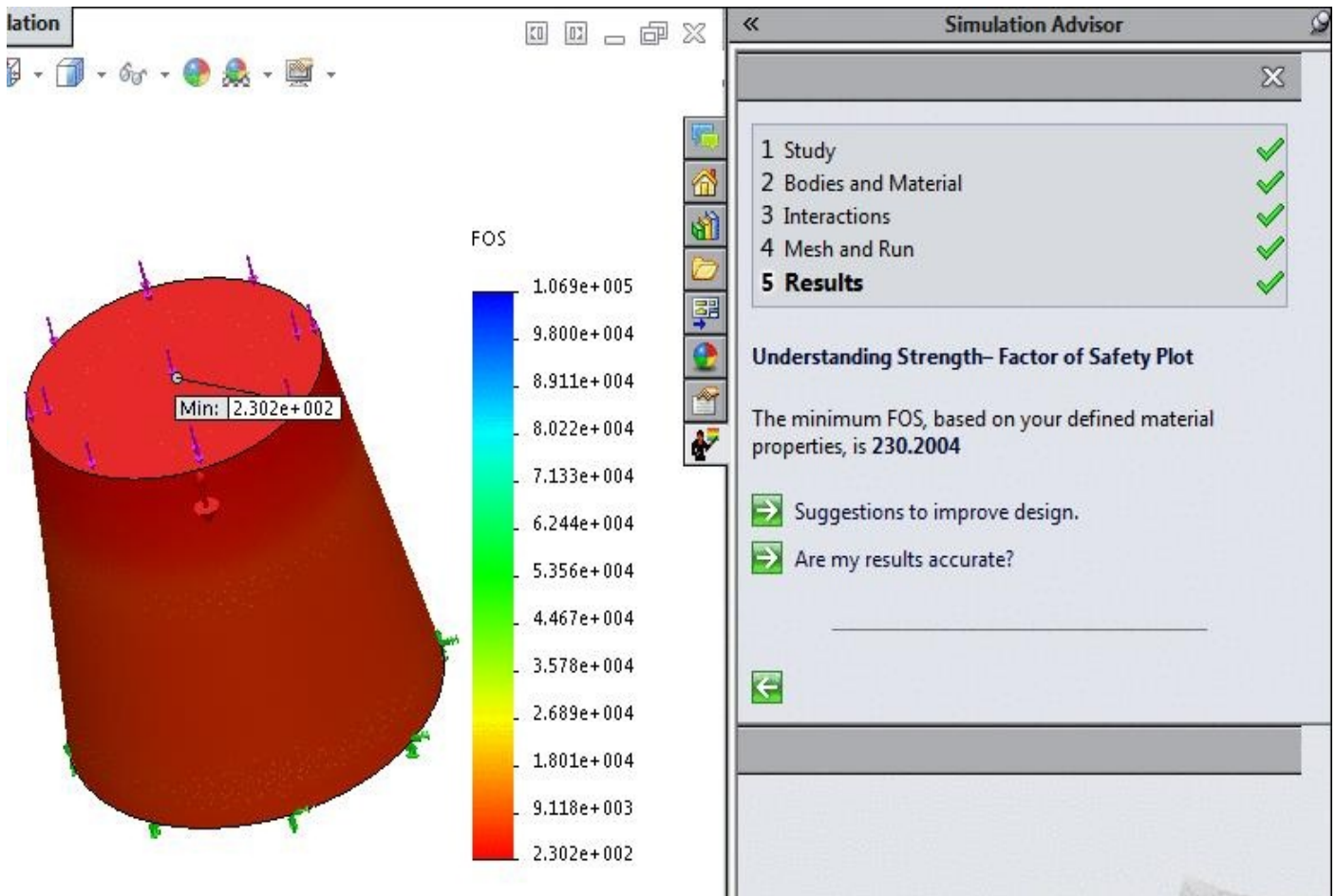


Figure-51. Factor of safety result

- Click on the **Suggestions to improve design** link button and you will get to know the methods to increase strength of the product if required; refer to [Figure-52](#).

Understanding Strength- Suggestions for Improvement

There are other common techniques for improving strength or reducing stress you may want to consider. Review each of these that you think might be applicable to your problem.

- Consider alternate material. [Example](#)
- Add or increase fillet radii.
- Add ribs.
- Add gussets.
- Increase number of fasteners.

Figure-52. Suggestions to improve strength

- By default, the Von-mises stress plot is displayed but sometimes we need other plots of stress also. To do so, right-click on the **Results** node in the **Analysis Manager** and select the **Define Stress Plot** option. The **Stress Plot PropertyManager** will be displayed; refer to [Figure-53](#).

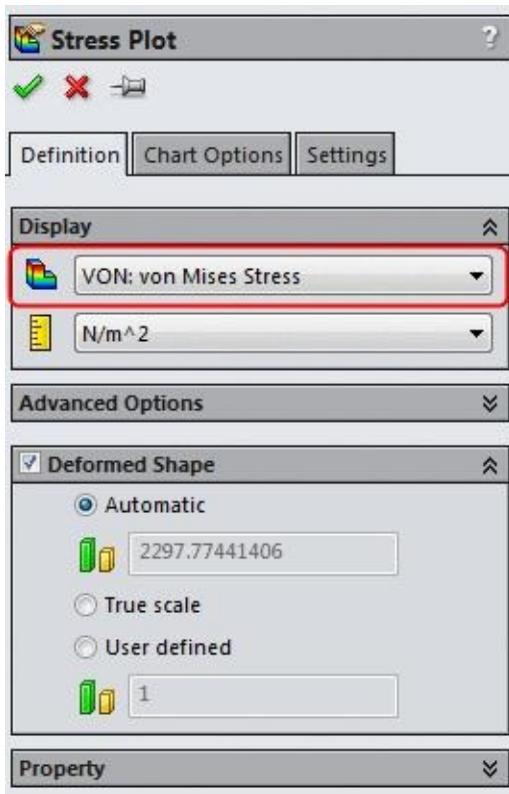


Figure-53. Stress Plot PropertyManager

- Click on the **Component** drop-down and select the **P1: 1st Principal Stress** option from the list displayed.
- Click on the **OK** button from the **PropertyManager**. The plot will be created.

PRACTICE 1

Consider a rectangular plate with cutout. The dimensions and the boundary conditions of the plate are shown in [Figure-54](#). It is fixed on one end and loaded on the other end. Under the given loading and constraints, plot the deformed shape. Also, determine the principal stresses and the von Mises stresses in the bracket. Thickness of the plate is 0.125 inch and material is **AISI 1020**.

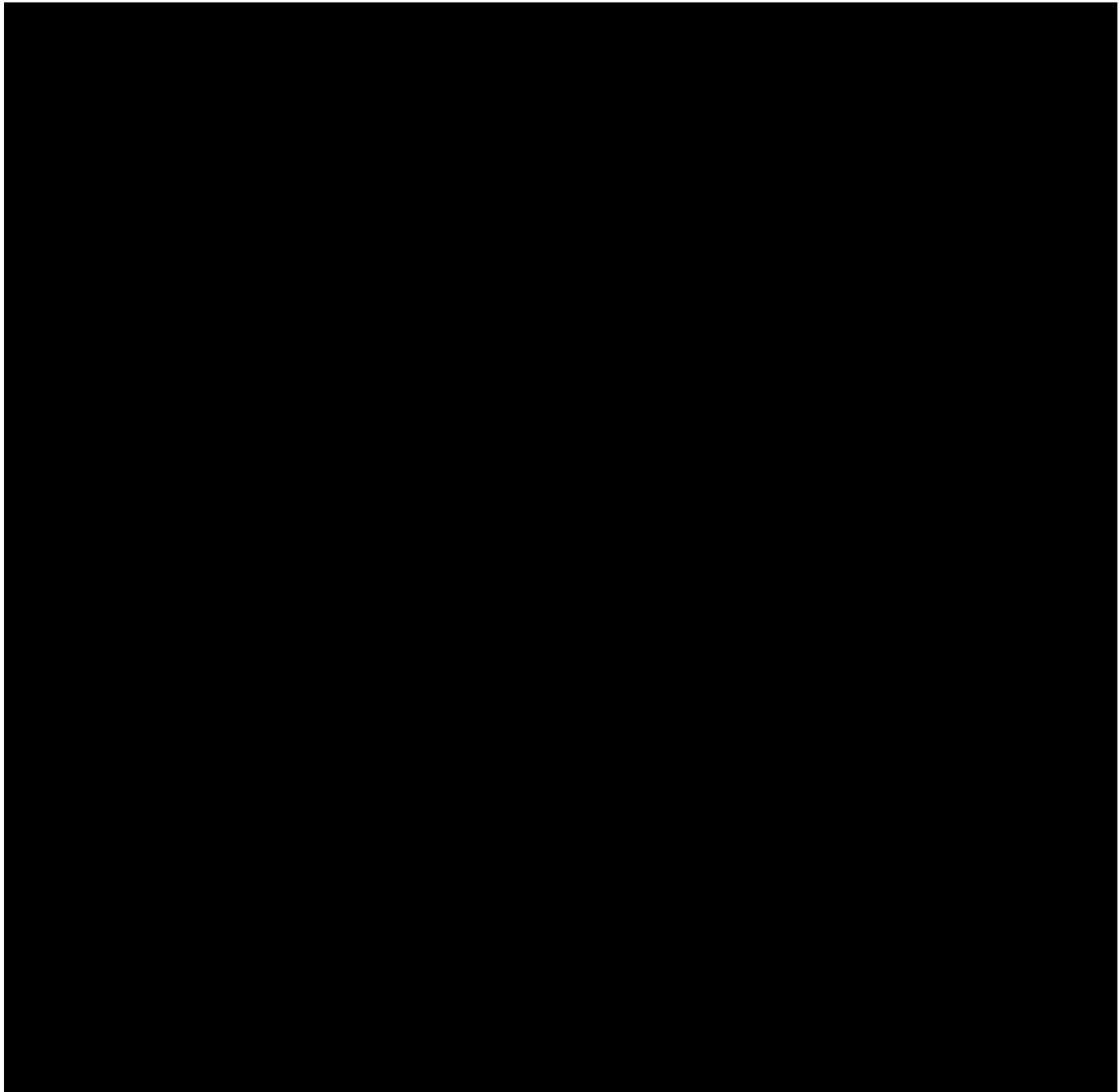


Figure-54. Drawing for Practice 1

P_RACTICE 2

Download the model for practice 2 of chapter 3 from the resource kit and perform the static analysis using the conditions given in [Figure-55](#). Find out the **Factor of Safety** for the model.

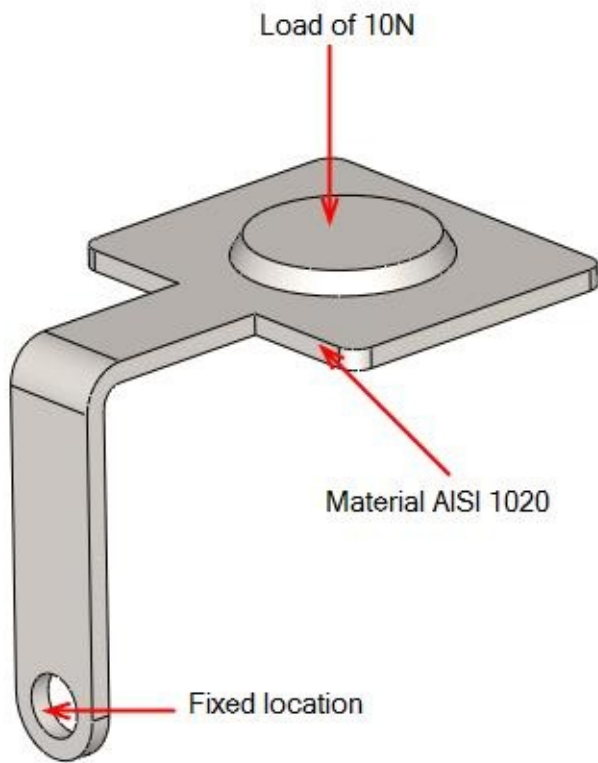


Figure-55. Practice 2

PRACTICE 3

Download the model of chair in chapter 3 from the resource kit and perform the static analysis using the conditions given in [Figure-56](#). The material used for manufacturing chair is PET (Polyethylene terephthalate). Check whether it is safe. Also, check whether Rikishi (wrestler) can sit on this chair without any consequences. His weight is 193 Kg.

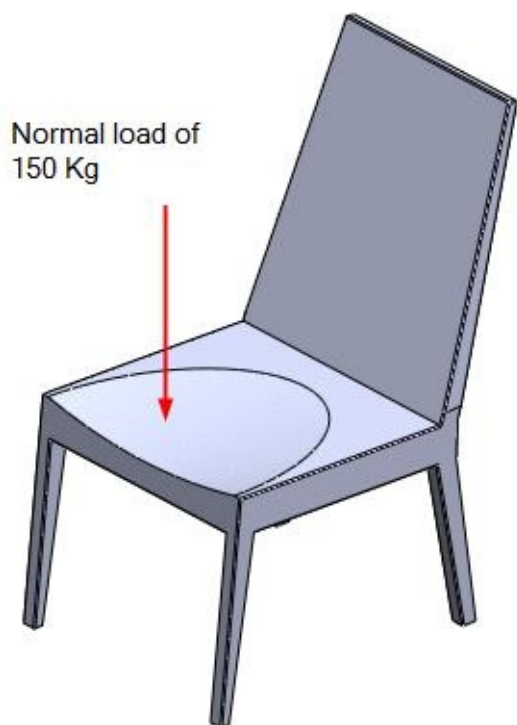


Figure-56. Practice 3 for Static Analysis

NON-LINEAR STATIC ANALYSIS

CHAPTER 4



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Non-Linear Static Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

NON LINEAR STATIC ANALYSIS

All real structures behave nonlinearly in one way or another at some level of loading. In some cases, linear analysis may be adequate. In many other cases, the linear solution can produce erroneous results because the assumptions upon which it is based are violated. Nonlinearity can be caused by the material behavior, large displacements, and contact conditions.

You can use a nonlinear study to solve a linear problem. The results can be slightly different due to different procedures.

In nonlinear finite element analysis, a major source of nonlinearity is due to the effect of large displacements on the overall geometric configuration of structures. Structures undergoing large displacements can have significant changes in their geometry due to load-induced deformations which can cause the structure to respond nonlinearly in a stiffening and/or a softening manner.

For example, cable-like structures generally display a stiffening behavior on increasing the applied loads while arches may first experience softening followed by stiffening, a behavior widely-known as the snap-through buckling.

Another important source of nonlinearity stems from the nonlinear relationship between the stress and strain which has been recognized in several structural behaviors. Several factors can cause the material behavior to be nonlinear. The dependency of the material stress-strain relation on the load history (as in plasticity problems), load duration (as in creep analysis), and temperature (as in thermoplasticity) are some of these factors.

This class of nonlinearity, known as material nonlinearity, can be idealized to simulate such effects which are pertinent to different applications through the use of constitutive relations.

A special class of nonlinear problems is concerned with the changing nature of the boundary conditions of the structures involved in the analysis during motion. This situation is encountered in the analysis of contact problems.

Pounding of structures, gear-tooth contacts, fitting problems, threaded connections, and impact bodies are several examples requiring the evaluation of the contact boundaries. The evaluation of contact boundaries (nodes, lines, or surfaces) can be achieved by using gap (contact) elements between nodes on the adjacent boundaries.

Yielding of beam-column connections during earthquakes is one of the applications in which material nonlinearity are plausible.

In this chapter, we are going to start with Non-linear Static Analysis which means there is no damping or resistance to the force.

Starting Nonlinear Static Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Nonlinear** button and then on the **Static** button from the **Options** rollout; refer to [Figure-1](#).

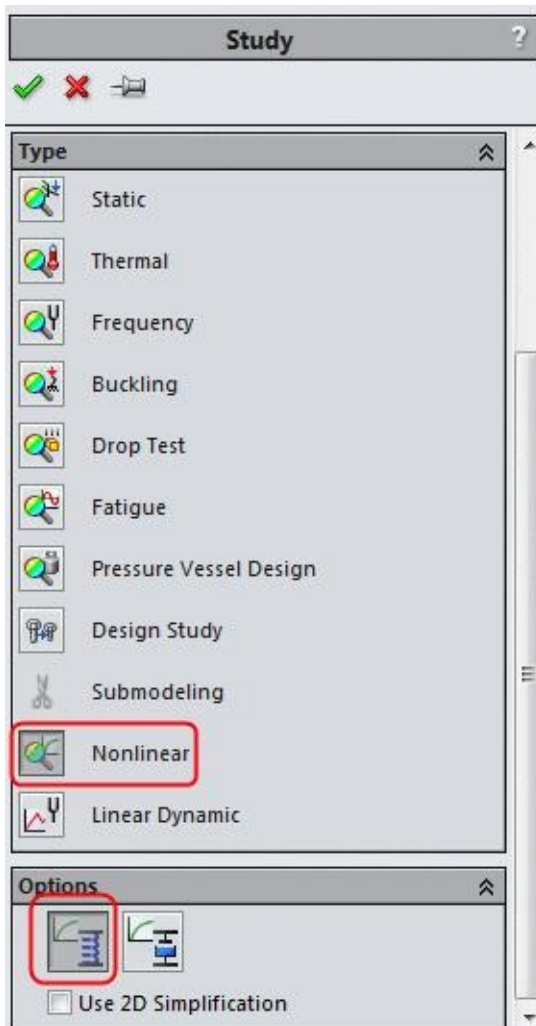


Figure-1. Study PropertyManager

- Click on the **OK** button from the **PropertyManager**. The tools related to **Non-linear Static** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

-

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed as discussed in earlier chapters.
- Select the **AISI 1035 Steel(SS)** option from the left list and click on the **Apply** button.
- Click on the **Close** button from the dialog box to exit. The material will be applied; refer to [Figure-2](#).

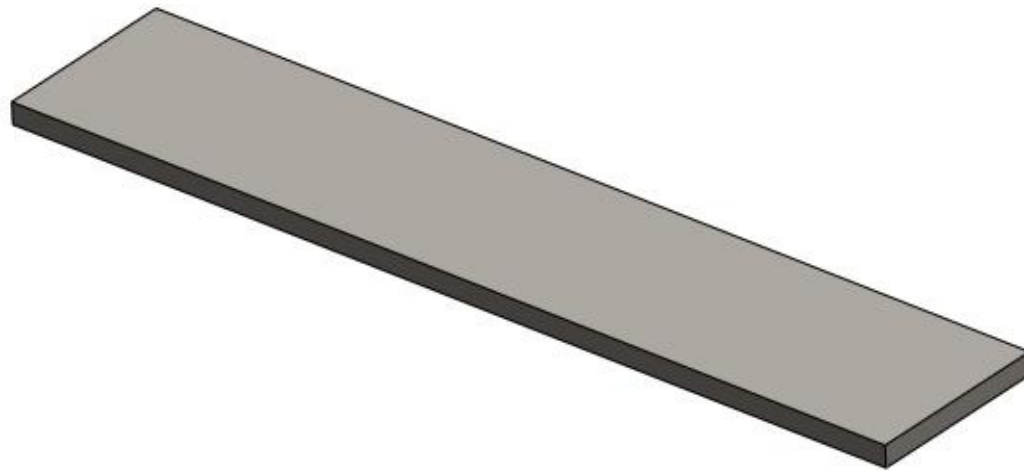


Figure-2. Model after applying material

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed; refer to [Figure-3](#).

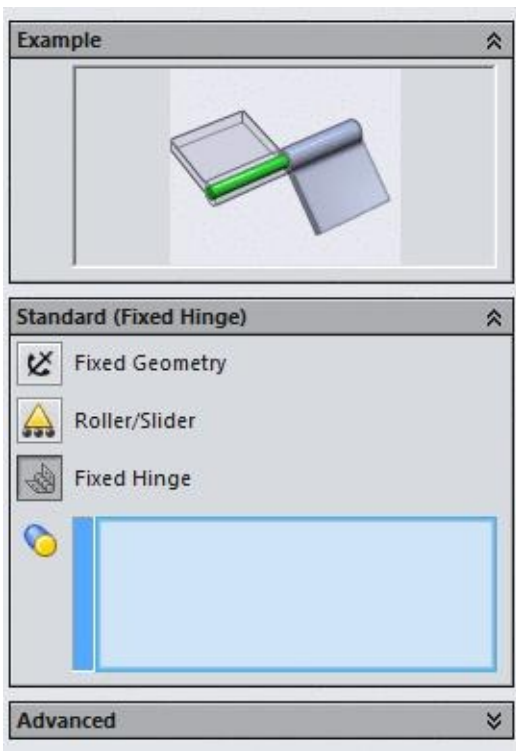


Figure-3. Fixture PropertyManager

- Select the left flat faces of the model; refer to [Figure-4](#).

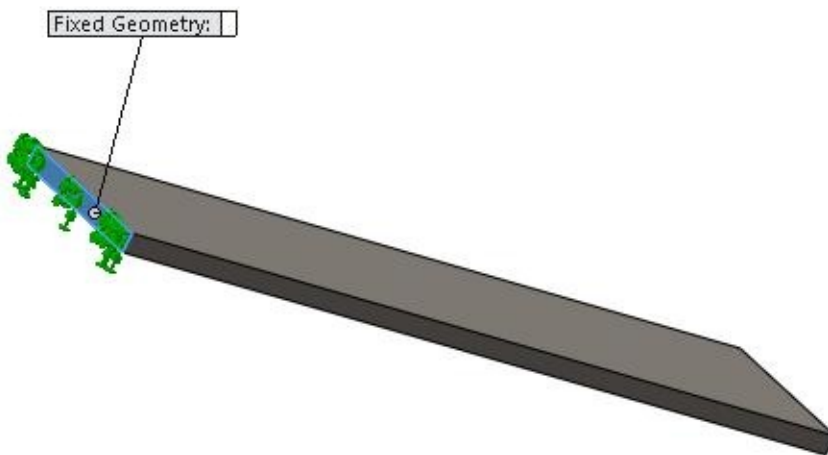


Figure-4. Faces fixed for analysis

- Click on the **OK** button from the **PropertyManager**. The holes will be fixed as if they are bolted at some position.

Applying Time Varying Force

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed; refer to [Figure-5](#).
- Note that **Variation with Time** rollout is added in the PropertyManager and two radio buttons are displayed named, **Linear** and **Curve**. On selecting the **Linear** radio button, the force will gradually increase till it reaches to the specified force value within the **1** second of time span; refer to [Figure-6](#).

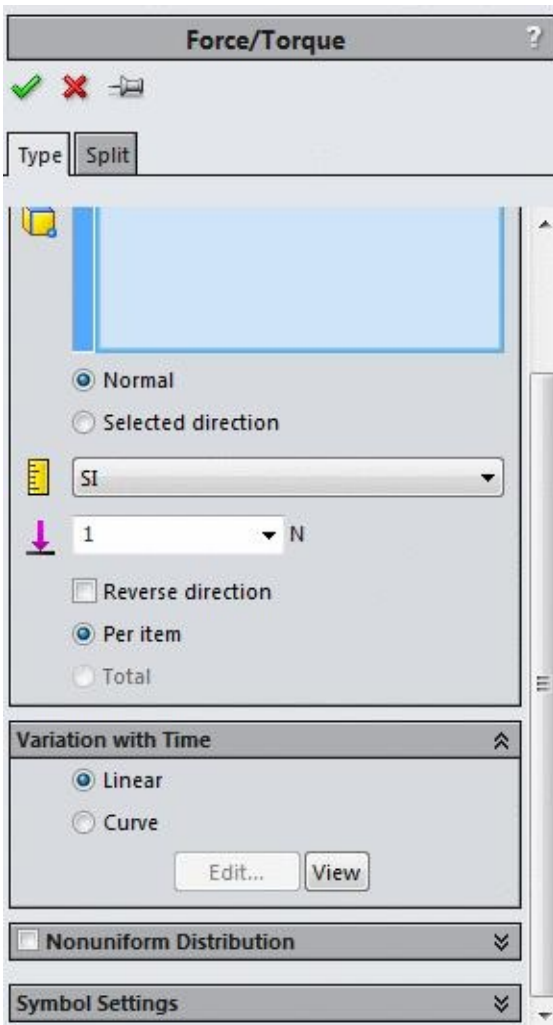


Figure-5. Force Torque PropertyManager

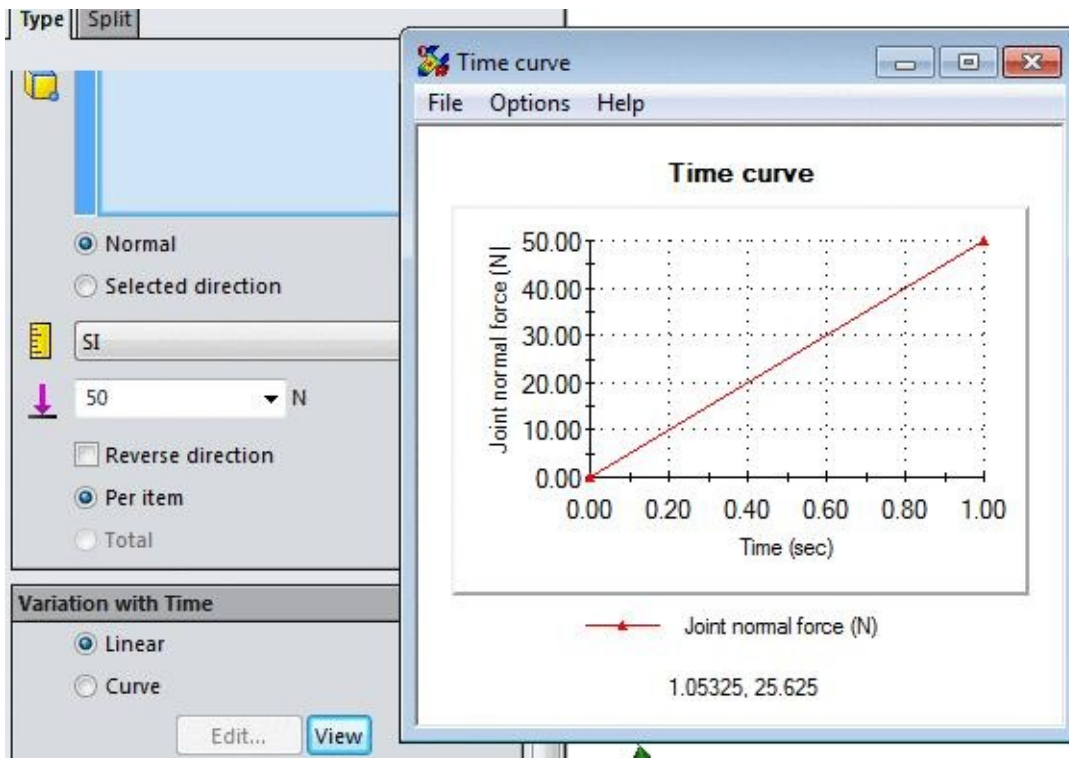


Figure-6. Time curve for linear variation

- Click on the **Curve** radio button from the **Variation with Time** rollout. The **Edit** button will become active.
- Click on the **Edit** button. The **Time curve** dialog box will be displayed as shown in [Figure-7](#).

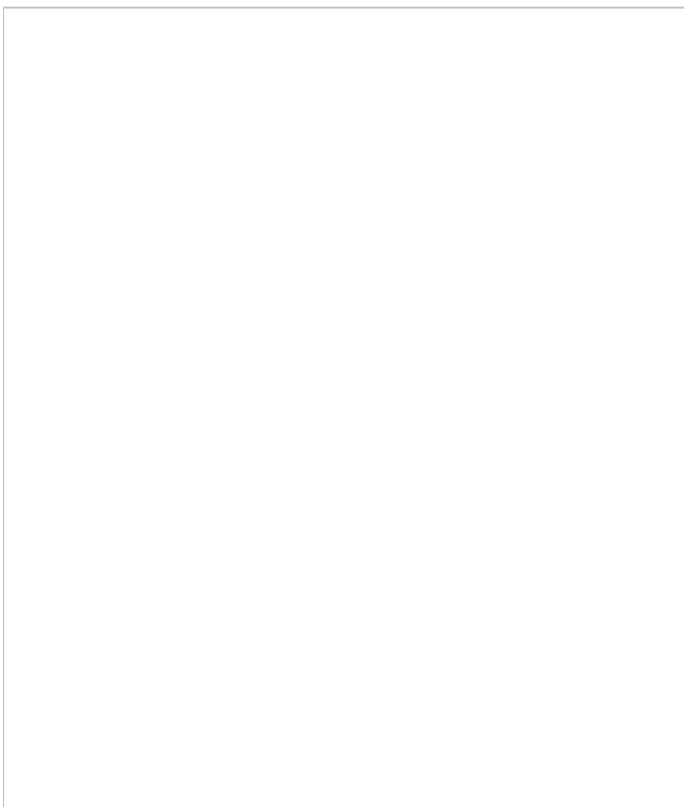
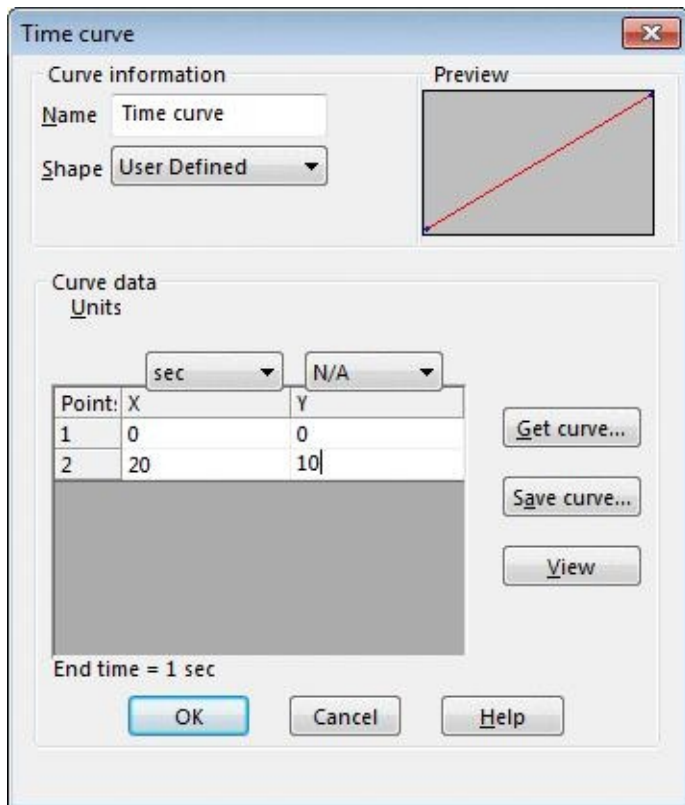


Figure-7. Time curve dialog box

- Click in the 2nd row of first column and specify the time as 20 in the field. In the same way, click in the 2nd row of the 2nd column and specify the force value as 10; refer to [Figure-8](#).

Figure-8. Time curve dialog box for customization



- Click on the OK button from the dialog box and then select the edge as shown in [Figure-9](#) to apply the force.

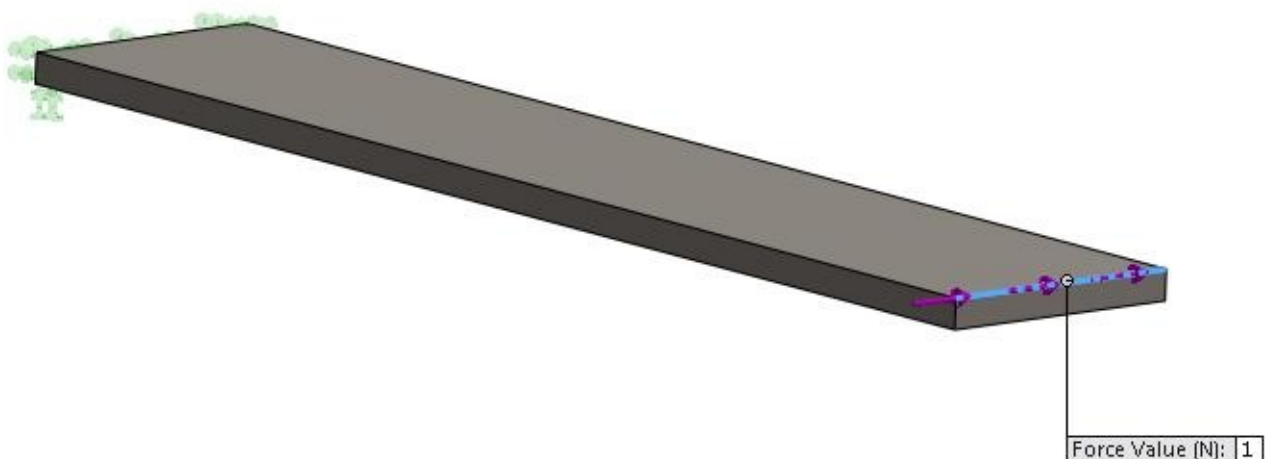


Figure-9. Edge selected for applying force

- Click on the **Selected direction** radio button from the **Force/Torque** rollout. You are asked to select a direction reference.
- Select the top face of the model; refer to [Figure-10](#).

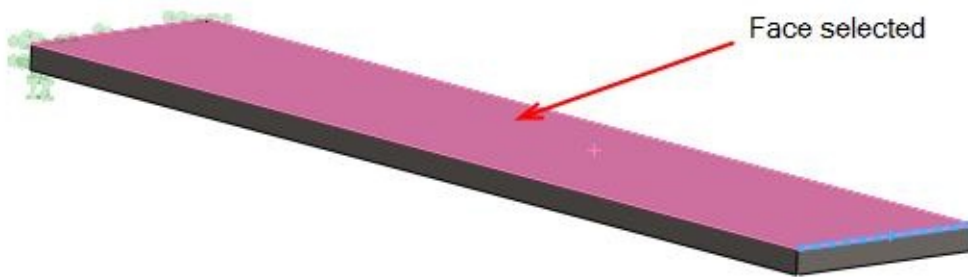


Figure-10. Face selected for force direction

- Now, click on the third button i.e. **Normal to Plane** and specify the force value as **10 N**; refer to [Figure-11](#).

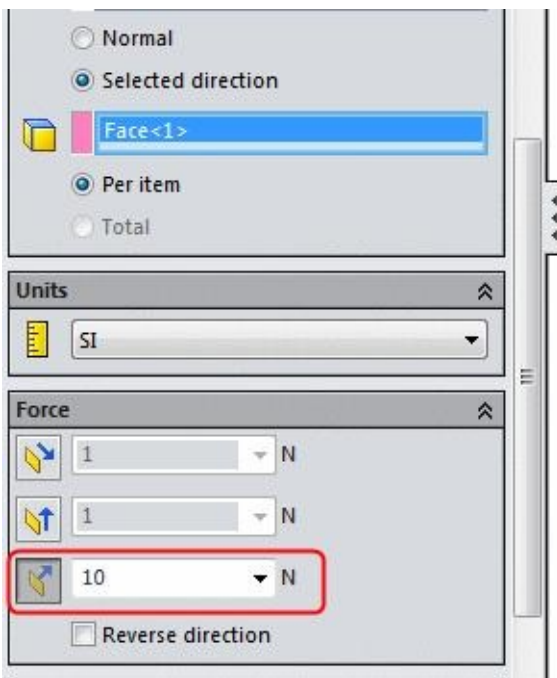


Figure-11. Button to be selected from Force rollout

- Note that if you click on the **View** button from the **Variation with Time** rollout; the graph will be displayed with the total force of $10 \times 10 = 100$ N; refer to [Figure-12](#).

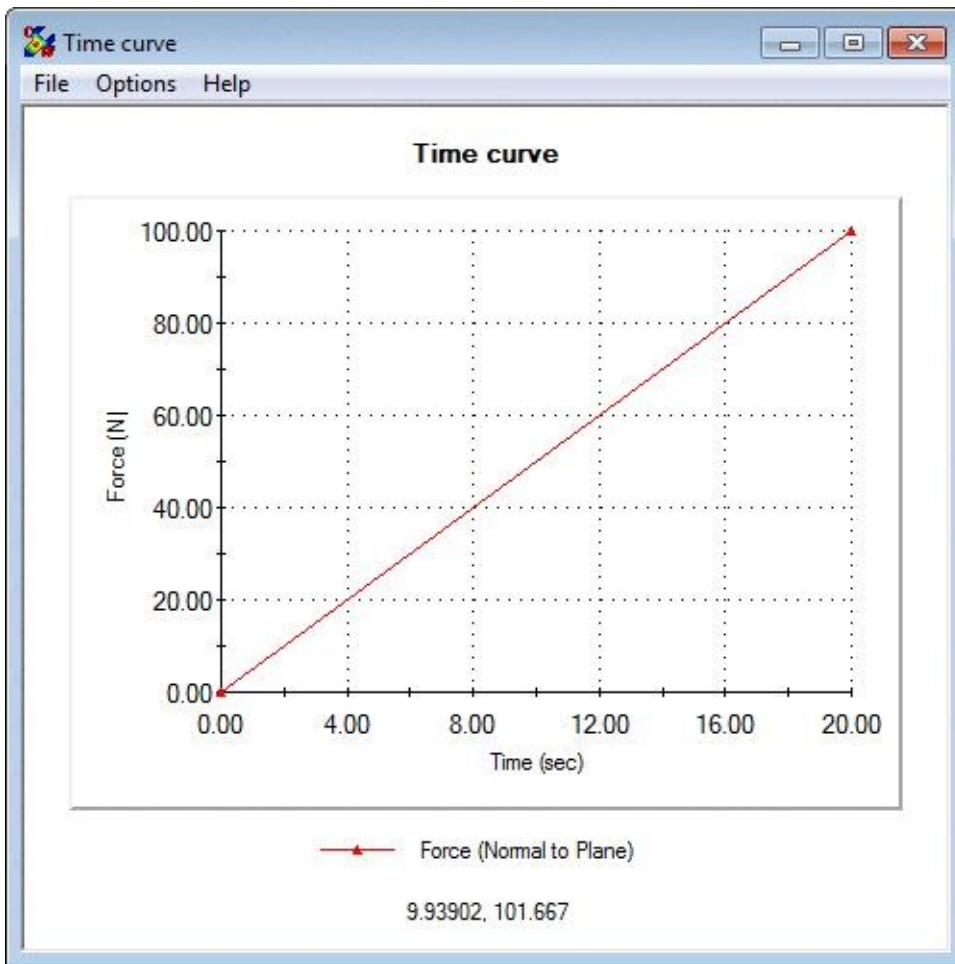


Figure-12. Time curve dialog box after entering 10 in force edit box

- Close the dialog box and click on the **OK** button from the **PropertyManager**. The force will be applied.

Running the analysis

- Click on the **Run** button from the **Ribbon**. The system will start analyzing the problem. Once the system is done, the results will be displayed in the Modeling area; refer to [Figure-13](#). Note that in SolidWorks 2015, you can also see the intermediate results by selecting the **Pause** button from the **Solver** dialog box. Make sure that you have selected the **Show intermediate results upto current step** check box to display the results; refer to [Figure-14](#). If you find that results are not as per expectation then click on the **Cancel** button and perform the changes.

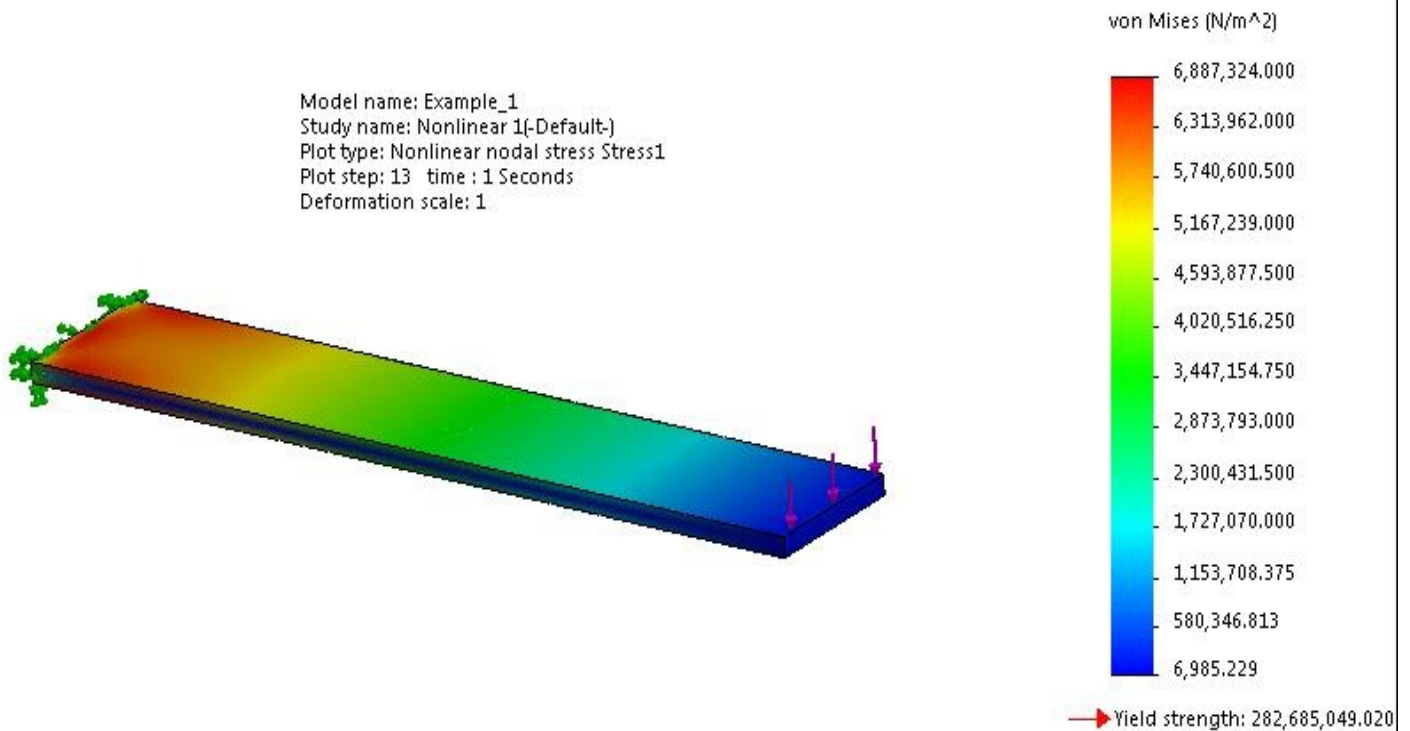


Figure-13. Result

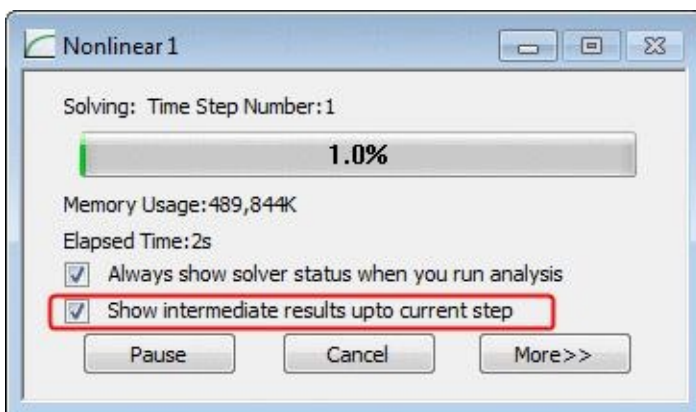


Figure-14. Non linear analysis solver dialog box

You can interpret the results as discussed in previous chapter.

PERFORMING NONLINEAR STATIC ANALYSIS ON AN ASSEMBLY

For performing non-linear static analysis on an assembly, we need to apply the contacts between various components in the assembly. The procedure to perform non-linear analysis on an assembly is given next.

Starting Non-Linear Static Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Nonlinear** button and then on the **Static** button from the **Options** rollout.
- Click on the **OK** button from the **PropertyManager**. The tools related to **Non-linear Static** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

Applying the Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed.
- Select the **Alloy Steel** material from the left and click on the **Apply** button.
- Click on the **Close** button from the dialog box. The material will be applied.

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed.
- Select the faces as shown in [Figure-15](#).

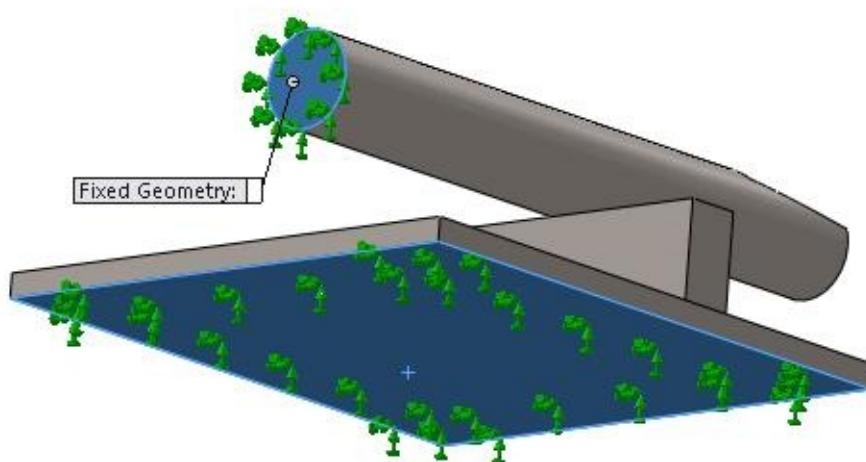


Figure-15. Faces selected for fixing

- Click on the **OK** button from the **PropertyManager** to fix the faces.

Applying Forces

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Note that **Variation with Time** rollout is added in the **PropertyManager** and two radio buttons are displayed named, **Linear** and **Curve**. Make sure that the **Linear** radio button is selected in the rollout.
- Enter the value of force as **10000** N in the **Force Value** edit box.
- Select the face as shown in [Figure-16](#).

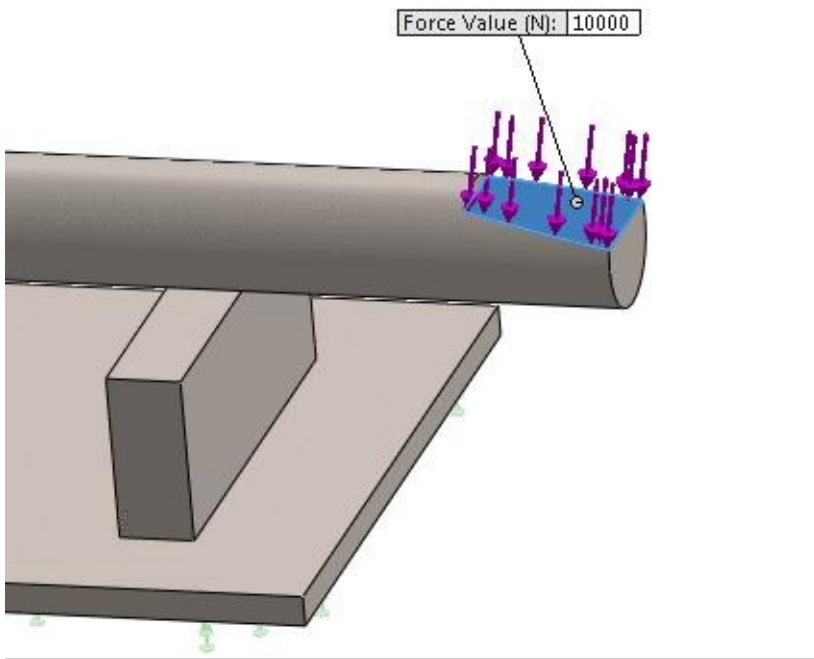


Figure-16. face selected for force

- Click on the **OK** button from the **PropertyManager** to apply the force.

Applying Connections

- Expand the **Component Contacts** node and right-click on the **Global Contact (-Bonded-)** option; refer to [Figure-17](#).

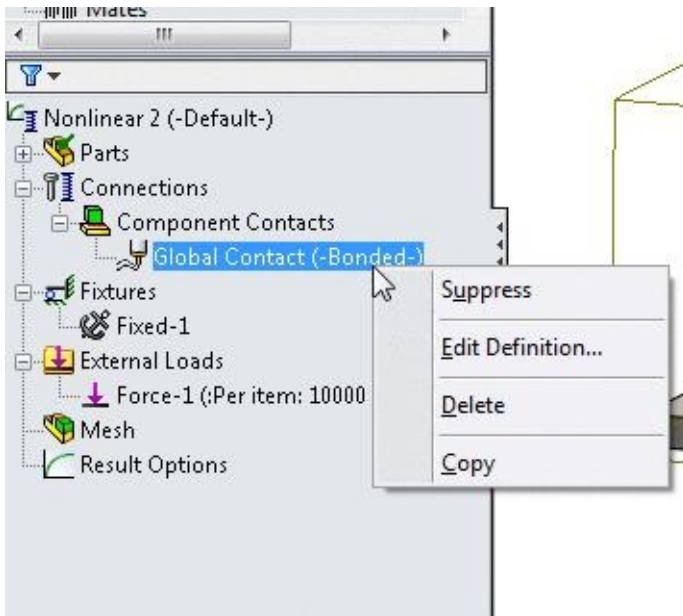


Figure-17. Option to be selected

- Select the **Delete** option from the shortcut menu. The Simulation dialog box will be displayed.
- Click on the **Yes** button to delete the contact.
- Click on the down arrow below the **Connections Advisor** button and select the **Component Contact** button; refer to [Figure-18](#).

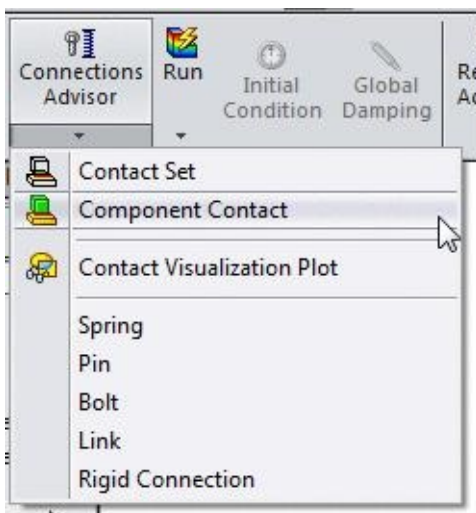


Figure-18. Component Contact button

- The **Component Contact PropertyManager** will be displayed as shown in [Figure-19](#).

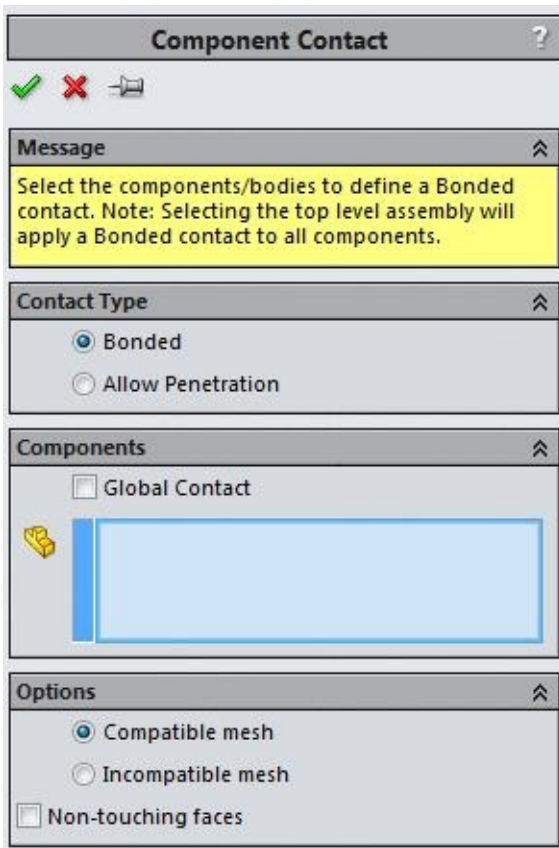


Figure-19. Component Contact PropertyManager

- Select the two components of the assembly as shown in [Figure-20](#).

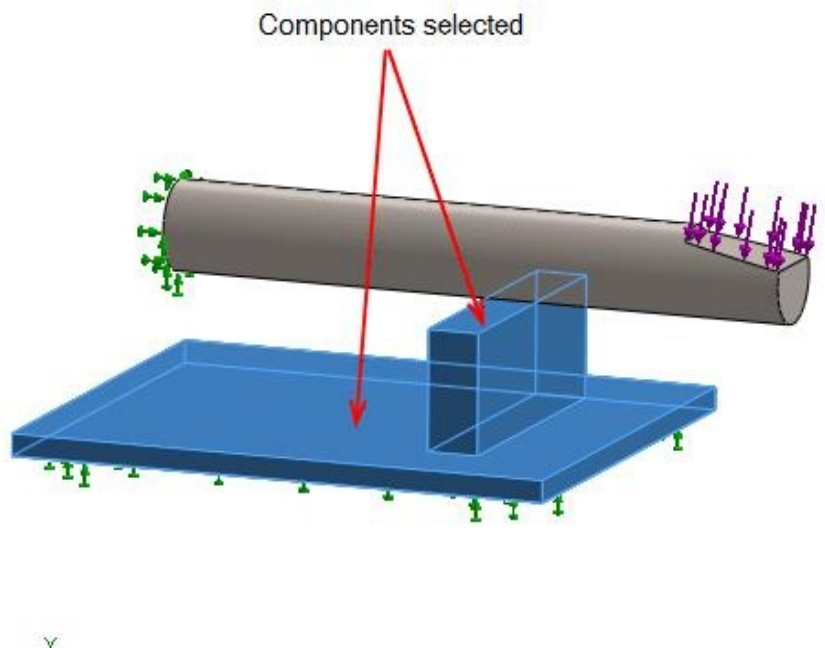
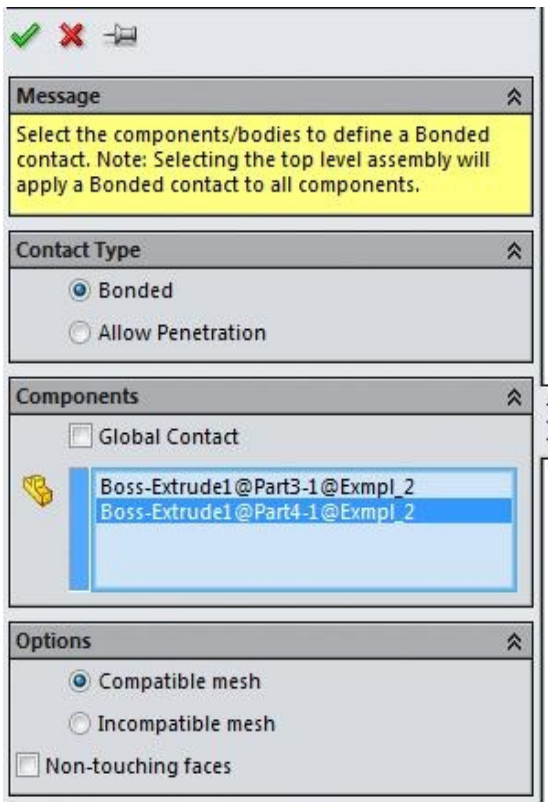


Figure-20. Components to be selected

- Click on the **OK** button from the **PropertyManager**. The contact will be added between

the two components.

- Again, click on the down arrow below the **Connections Advisor** button and click on the **Contact Set** button. The **Contact Sets PropertyManager** will be displayed as shown in [Figure-21](#).

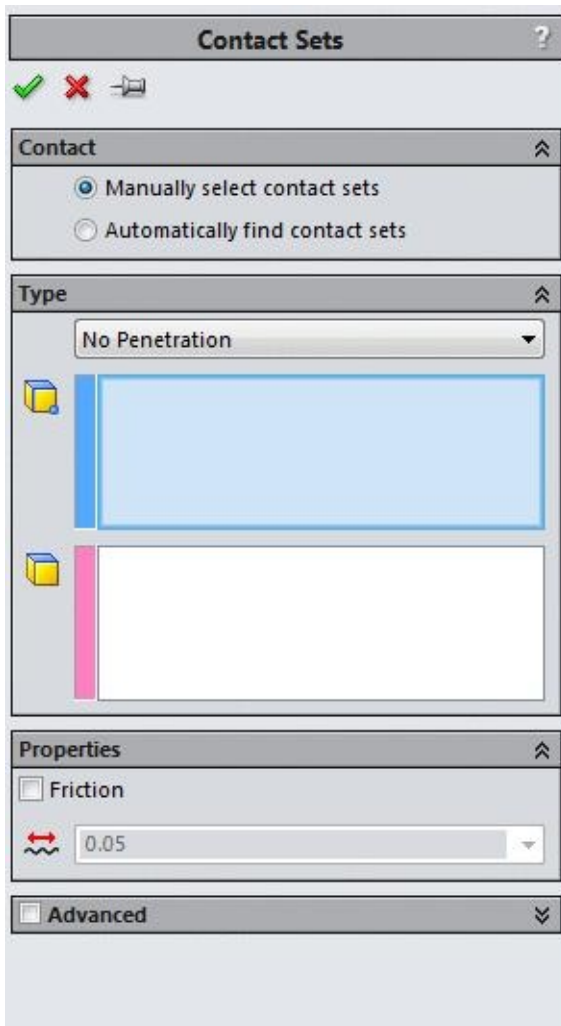


Figure-21. Contact Sets PropertyManager

- Select the round face of the first part in assembly.
- Click in the pink box in the **Type** rollout and select the flat faces of the block; refer to [Figure-22](#).

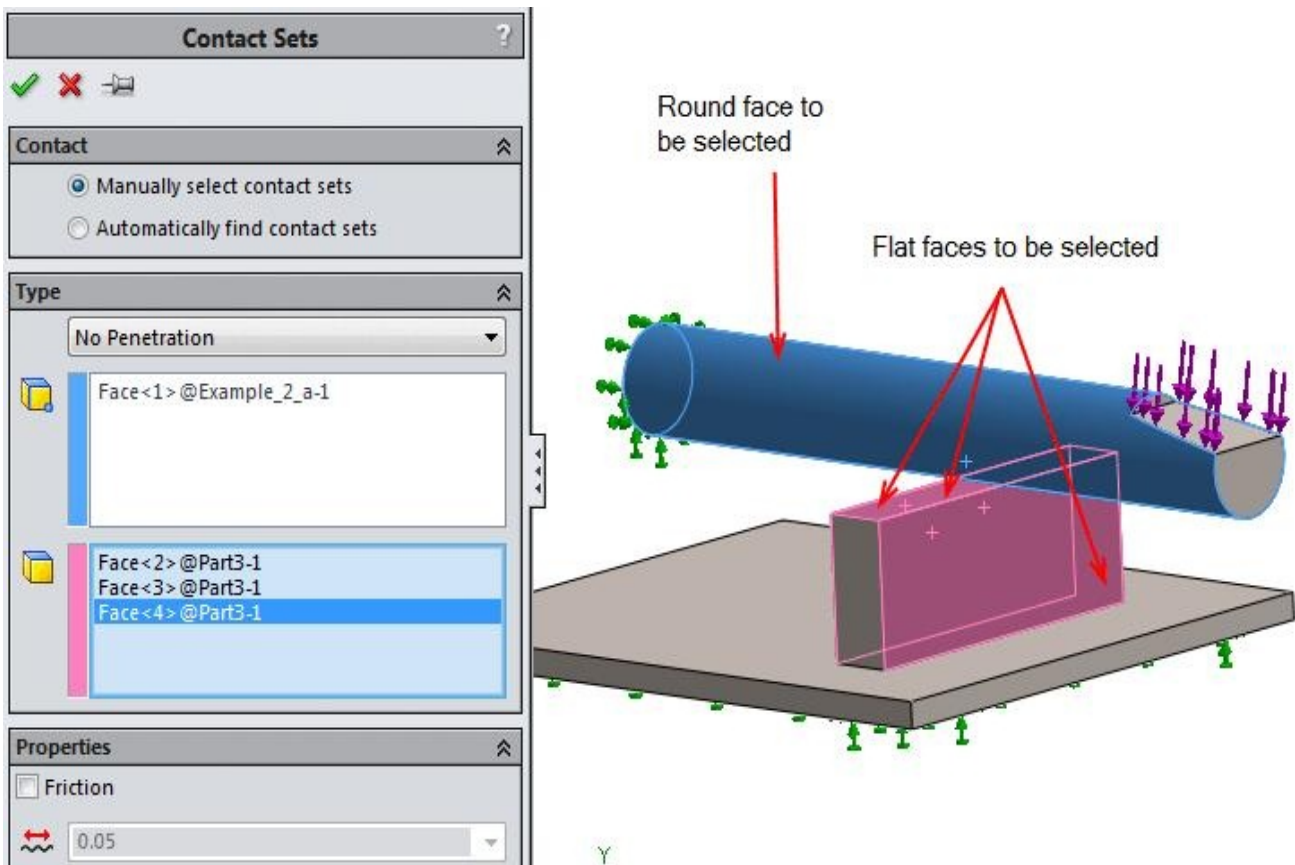


Figure-22. Faces selected for contact sets

- Select the check box next to *Advanced* rollout. The rollout will expand.
- Select the **Surface to surface** radio button.
- Click on the **OK** button from the **PropertyManager**. The contacts will be applied.

Running the analysis

- Click on the **Run** button from the **Ribbon**. The analysis will start.
- Note that since the faces that we selected later are not in contact to each other, so the dialog box with the related message; refer to [Figure-23](#).

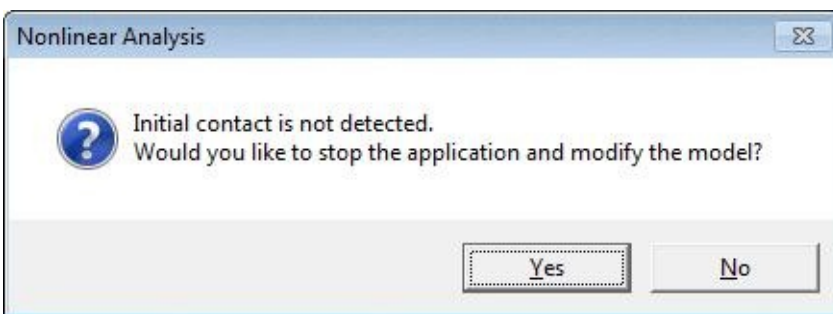


Figure-23. Nonlinear Analysis dialog box

- Click on the **No** button from the dialog box since we want to run the analysis with these conditions only. System will start solving the analysis.

- On completion of the analysis, the result will be displayed; refer to [Figure-24](#).

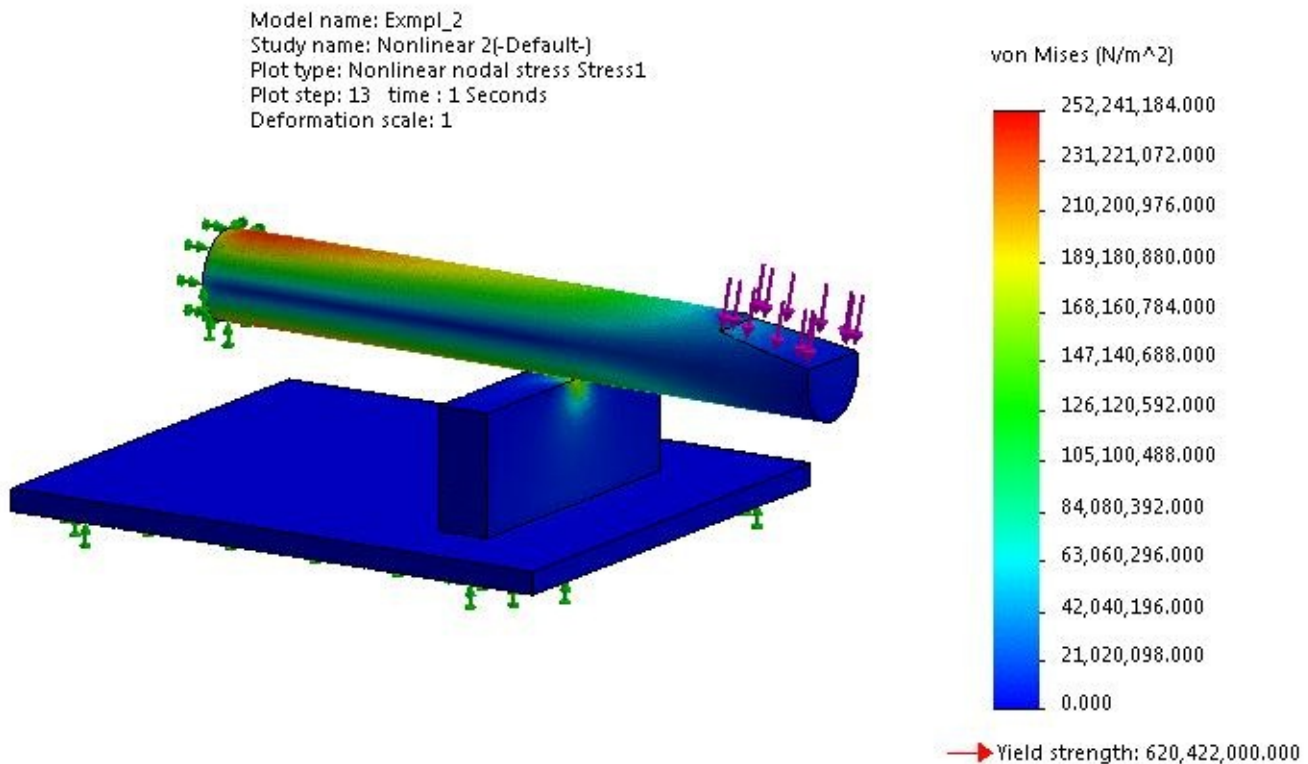


Figure-24. Analysis result

- Note that the stress is also induced in the block below the beam because of the force induced on the beam.
- You can change the contact between the surface of the beam and block to **Allow Penetration** using the **Contact Sets** button and check for the results for your interest.
- To animate any of the result, right-click on the result name from the **Analysis Manager** and select the **Animate** button from the shortcut menu; refer to [Figure-25](#).

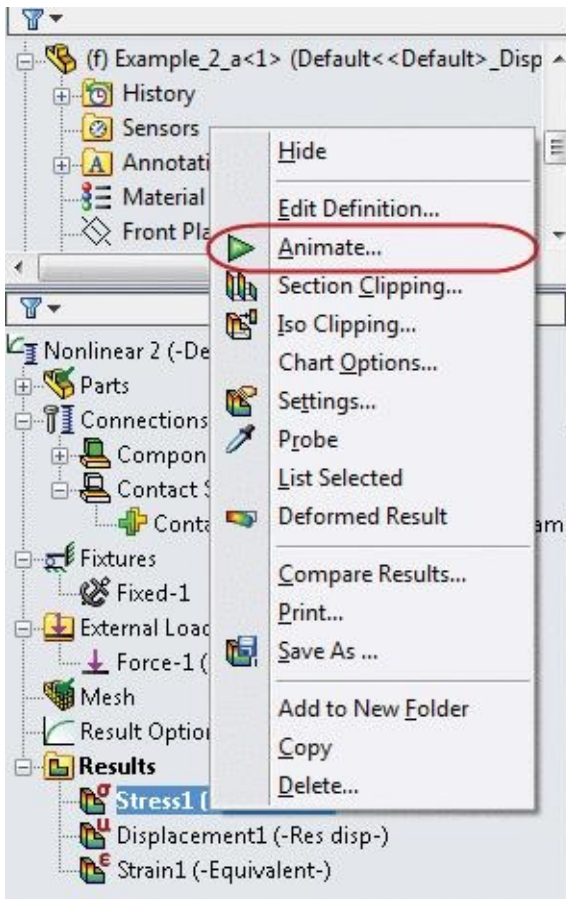


Figure-25. Animate button

- A simulation will be played for the current selected result.

PERFORMING NONLINEAR STATIC ANALYSIS ON A RING UNDER APPLICATION OF TORQUE

The model used in this example is a ring of cast iron under the load of a torque. The steps to perform the analysis are given next.

Starting Non-Linear Static Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Nonlinear** button and then on the **Static** button from the **Options** rollout.

- Click on the **OK** button from the **PropertyManager**. The tools related to **Non-linear Static** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

Applying the Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed.
- Select the **Alloy Steel** material from the left and click on the **Apply** button.
- Click on the **Close** button from the dialog box. The material will be applied.

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed.
- Select the faces as shown in [Figure-26](#). Click on the **OK** button to apply fixture.

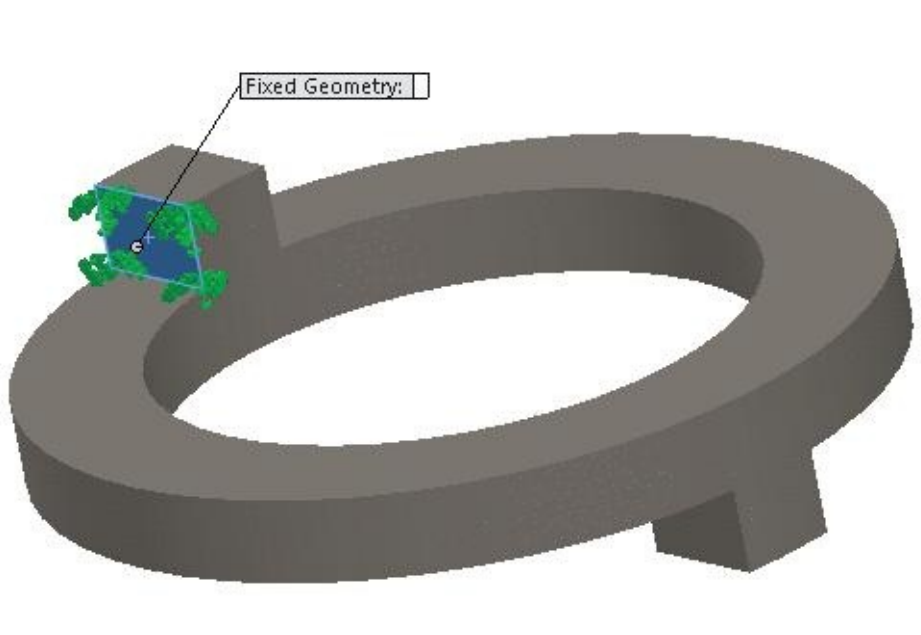


Figure-26. Face selected for fixture

- Since this is a round part and can deviate from its axis, so we need to restrict movement of its axis.
- Click again on the down arrow below the **Fixtures Advisor** button and select the **Fixed Hinge** button from the tool list. The **Fixture PropertyManager** will be displayed; refer to [Figure-27](#).

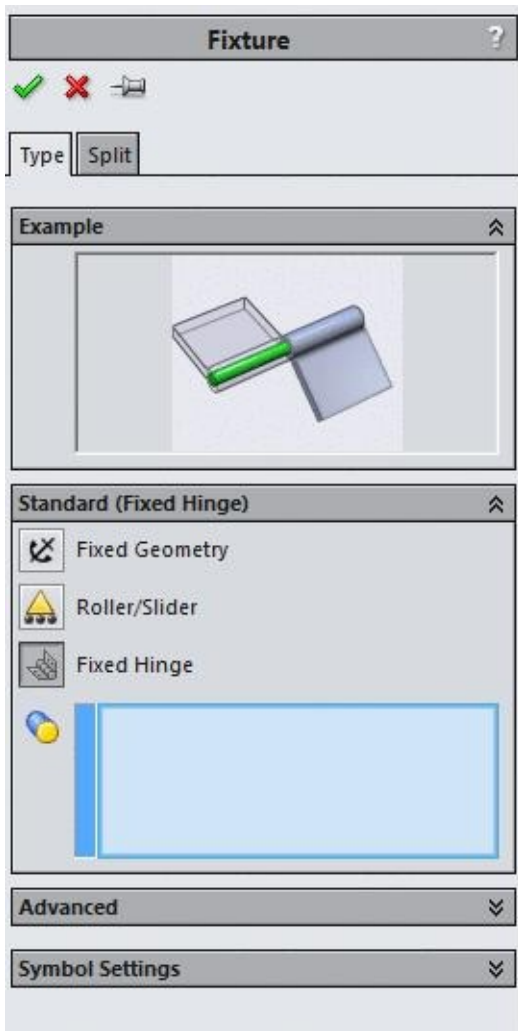


Figure-27. Fixture PropertyManager

- Select the inner round face of the model to make is fixed hinge; refer to [Figure-28](#).

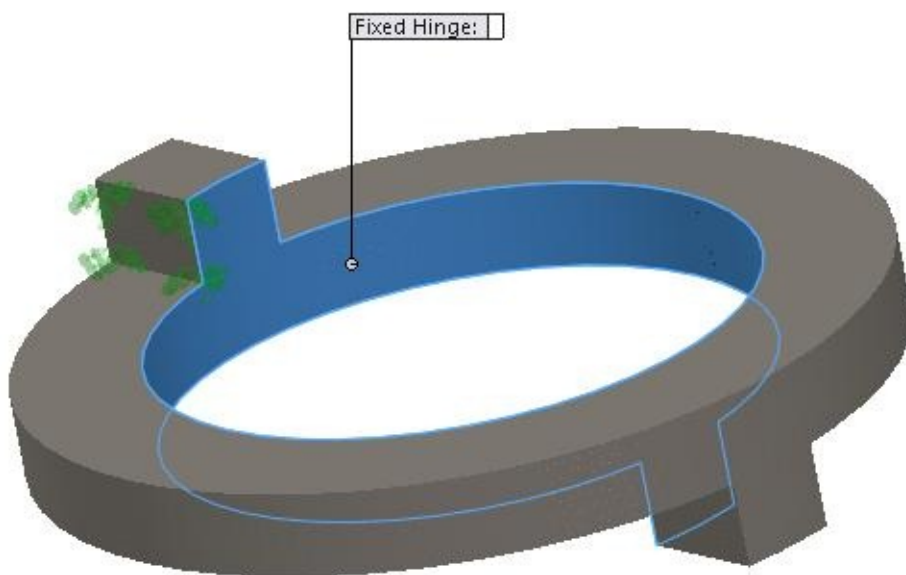


Figure-28. Round face selected for fixed hinge

- Click on the **OK** button from the **PropertyManager** to create the fixed hinge.

Applying Forces

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.
- Click on the **Torque** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Note that **Variation with Time** rollout is added in the **PropertyManager** and two radio buttons are displayed named, **Linear** and **Curve**. Make sure that the **Linear** radio button is selected in the rollout.
- Enter the value of force as **1000 N** in the **Force Value** edit box.
- Select the face as shown in [Figure-29](#).

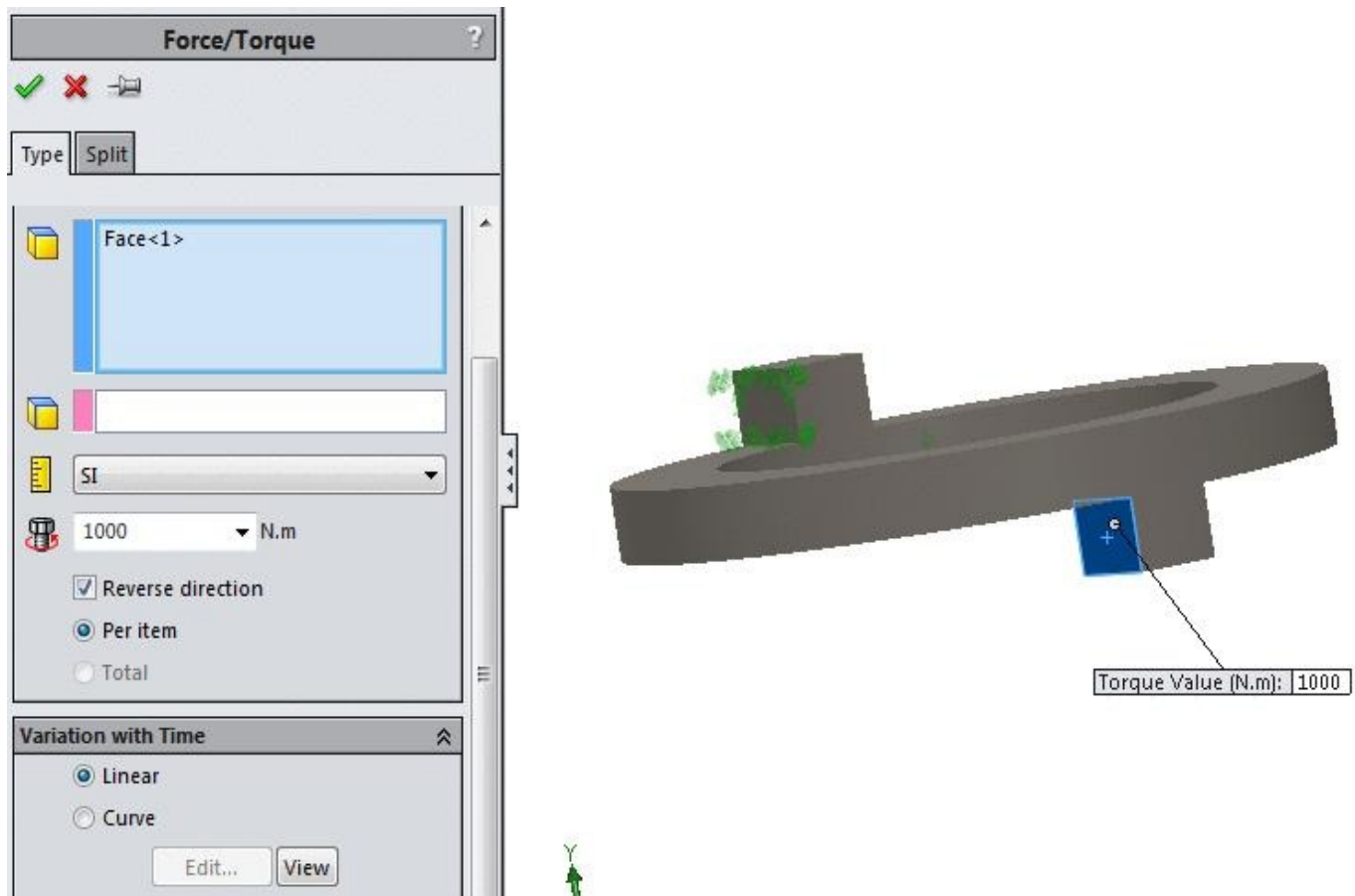


Figure-29. Face selected for applying torque

- Click in the pink box in **PropertyManager** and select the inner round face that was selected for fixed hinge. Preview of the torque will be selected.
- If change of torque direction is required then click on the **Reverse direction** check box. Click on the **OK** button from the PropertyManager to apply the torque.

Running the analysis

- Click on the **Run** button from the **Ribbon**. The analysis will start.
- After the processing is complete, the results will be displayed in the modeling area; refer to [Figure-30](#).

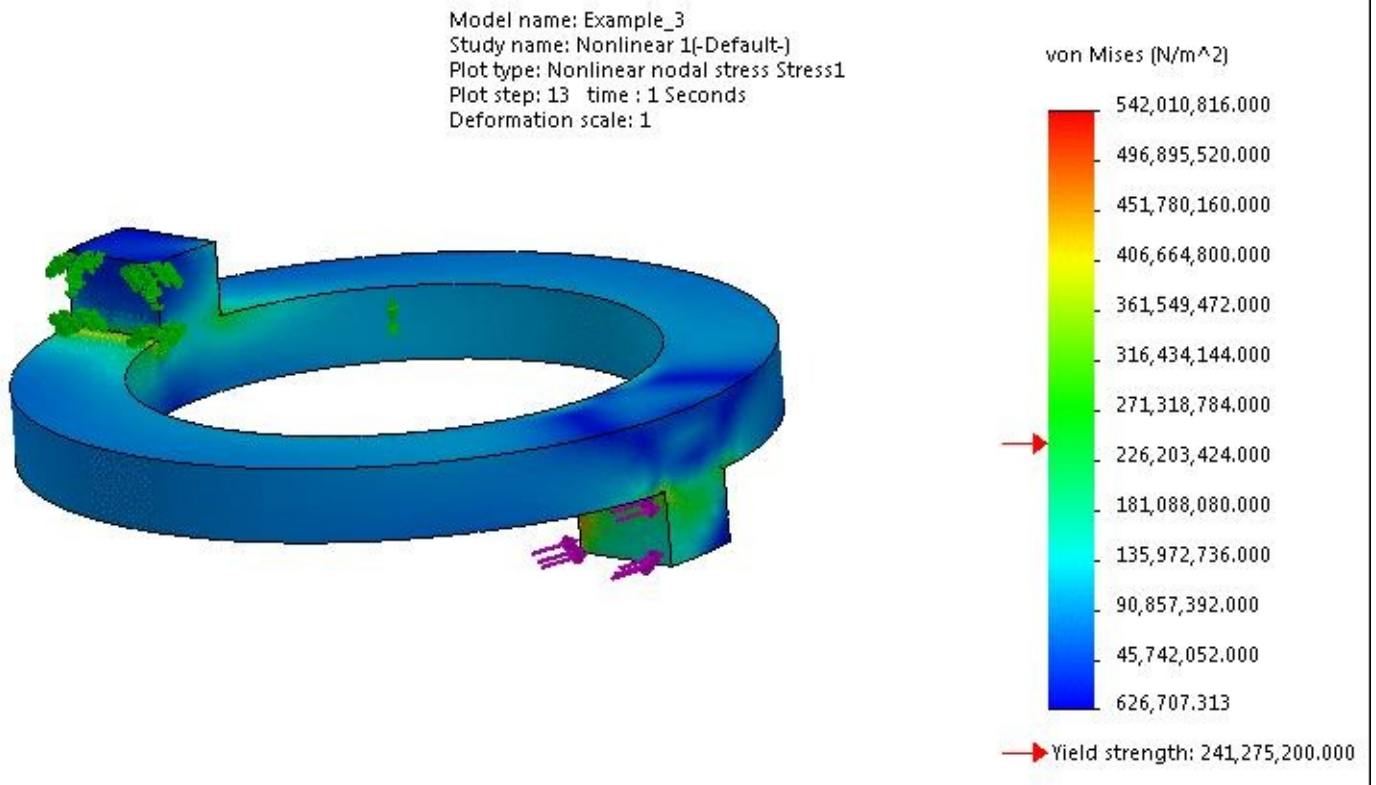


Figure-30. Result on applying torque

PRACTICE 1

Check out what happens to the fork under the conditions specified in [Figure-31](#). Note that the material of fork is **Alloy Steel** and it is in the hands of a nasty kid. (You know kids!! they don't exert linear forces.)

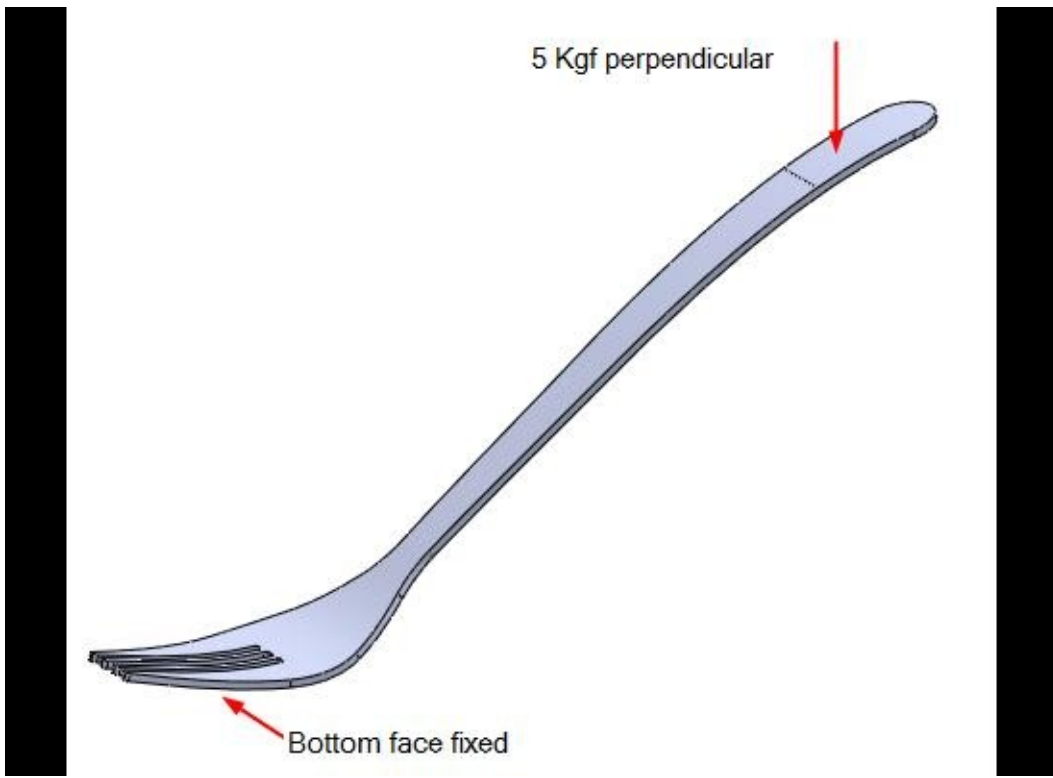


Figure-31. Practice 1 Non Linear Static Analysis

NON-LINEAR DYNAMIC ANALYSIS

CHAPTER 5



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Non-Linear Dynamic Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis*
- *Interpreting results*
- *Response Graphs*

NON LINEAR DYNAMIC ANALYSIS

In linear static analysis, the loads are applied gradually and slowly until they reach their full magnitude. After reaching their full magnitude, the loads remain constant (time-invariant). The accelerations and velocities of the excited system are negligible, therefore, no inertial and damping forces are considered in the formulation:

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

where:

[K]: stiffness matrix

{u}: displacement vector

{f}: load vector

The solution produces displacements, stresses, that are constant.

In linear dynamic analysis, the applied loads are time-dependent. The loads can be deterministic (periodic, non-periodic), or non-deterministic which means that they cannot be precisely predicted but they can be described statistically. The accelerations and velocities of the excited system are significant, therefore, inertial and damping forces should be considered in the formulation:

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = \{f(t)\}$$

where:

[K]: stiffness matrix

[C]: damping matrix

[M]: mass matrix

$\{u(t)\}$: time varying displacement vector

$\{\ddot{u}(t)\}$: time varying acceleration vector

$\{\dot{u}(t)\}$: time varying velocity vector

$\{f(t)\}$: time varying load vector

The response of the system is given in terms of time histories (amplitudes versus time), or in terms of frequency spectra (peak values versus frequency).

For linear dynamic analysis, the mass, stiffness, and damping matrices do not vary with time. Can you guess for what type of analysis they change with the time?

Starting Nonlinear Dynamic Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Nonlinear** button and then on the **Dynamic** button from the **Options** rollout; refer to [Figure-1](#).

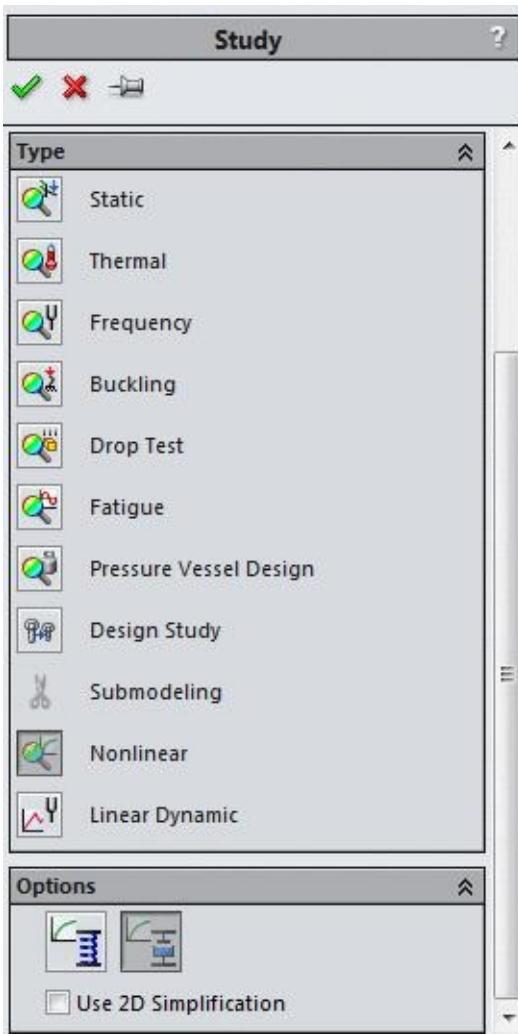


Figure-1. Study PropertyManager

- Click on the **OK** button from the **PropertyManager**. The tools related to **Non-linear Dynamic** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed as discussed in earlier chapters.
- Select the desired material, say **Alloy Steel** option from the left list and click on the **Apply** button.
- Click on the **Close** button from the dialog box to exit. The color of model will be changed; refer to [Figure-2](#).

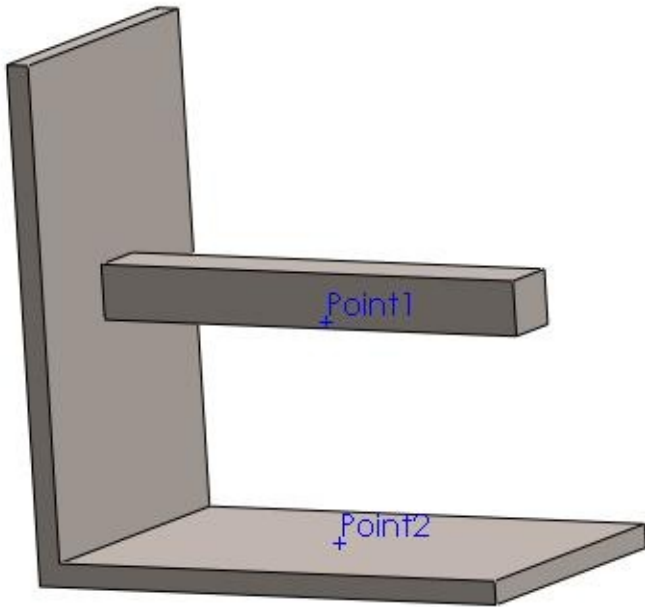


Figure-2. Model after applying material

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed; refer to [Figure-3](#).

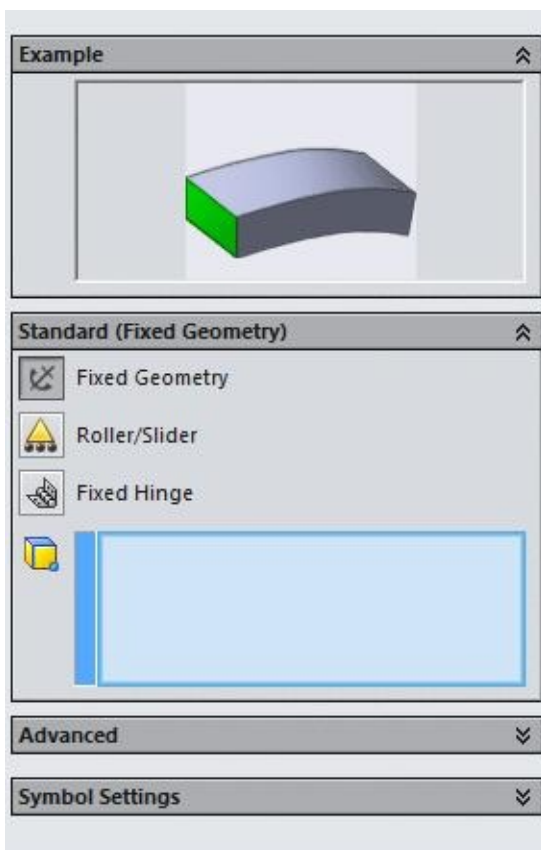


Figure-3. Fixture PropertyManager

- Select the flat faces of the model; refer to [Figure-4](#).

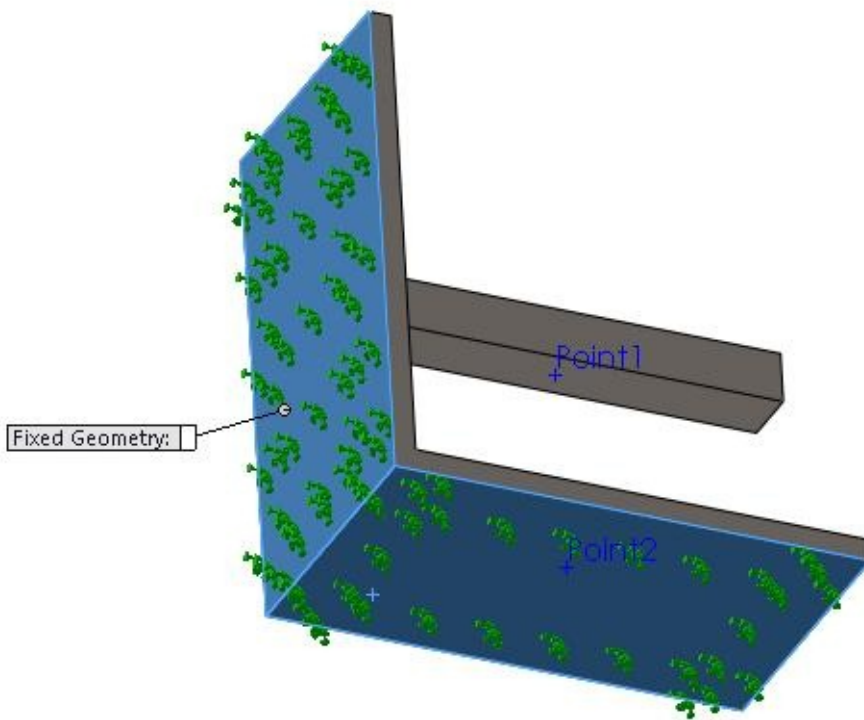


Figure-4. Faces to be fixed

- Click on the **OK** button from the **PropertyManager**. The holes will be fixed as if they are bolted at some position.

Applying Time Varying Force

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed; refer to [Figure-5](#).

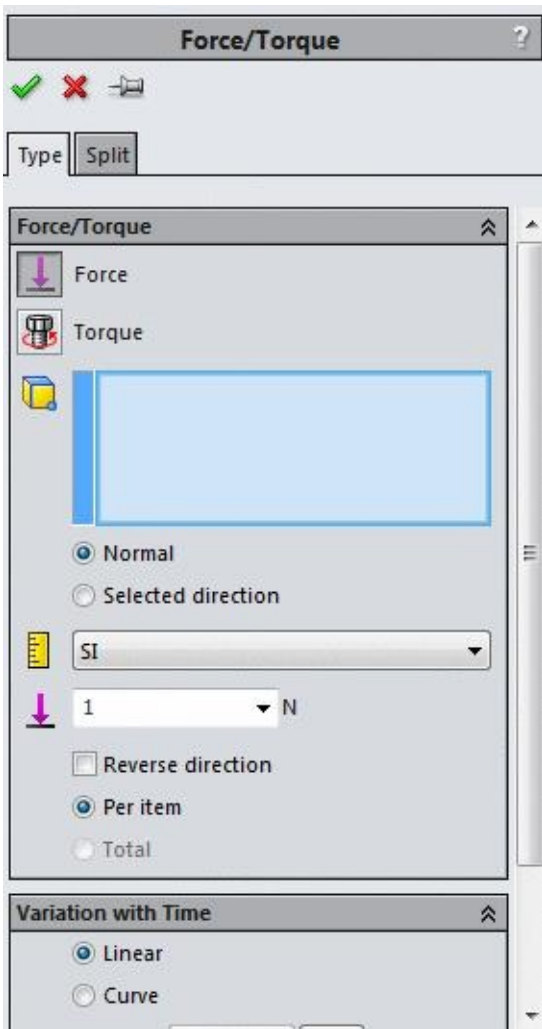


Figure-5. Force Torque PropertyManager

- Click on the edge as shown in [Figure-6](#).

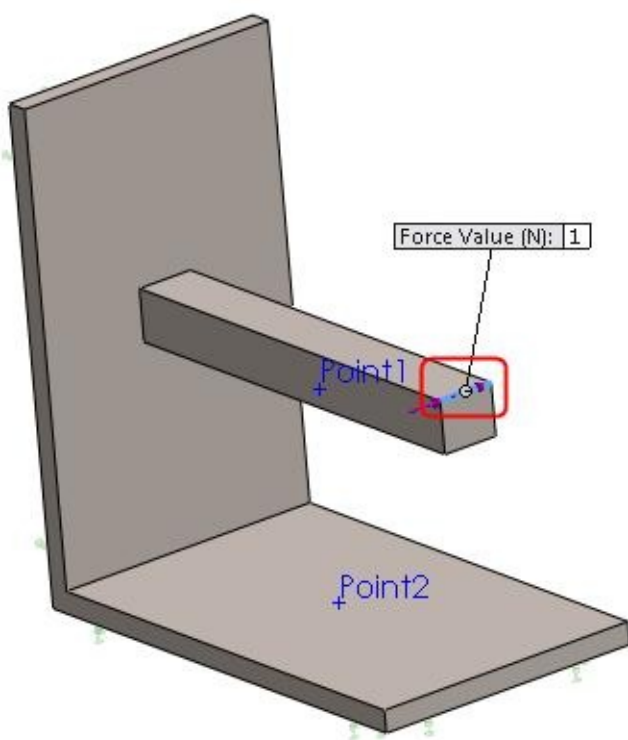


Figure-6. Edge to be selected

- Click on the **Selected direction** radio button. The **Force** rollout will be displayed.
- Select the top face of the model as shown in [Figure-7](#).

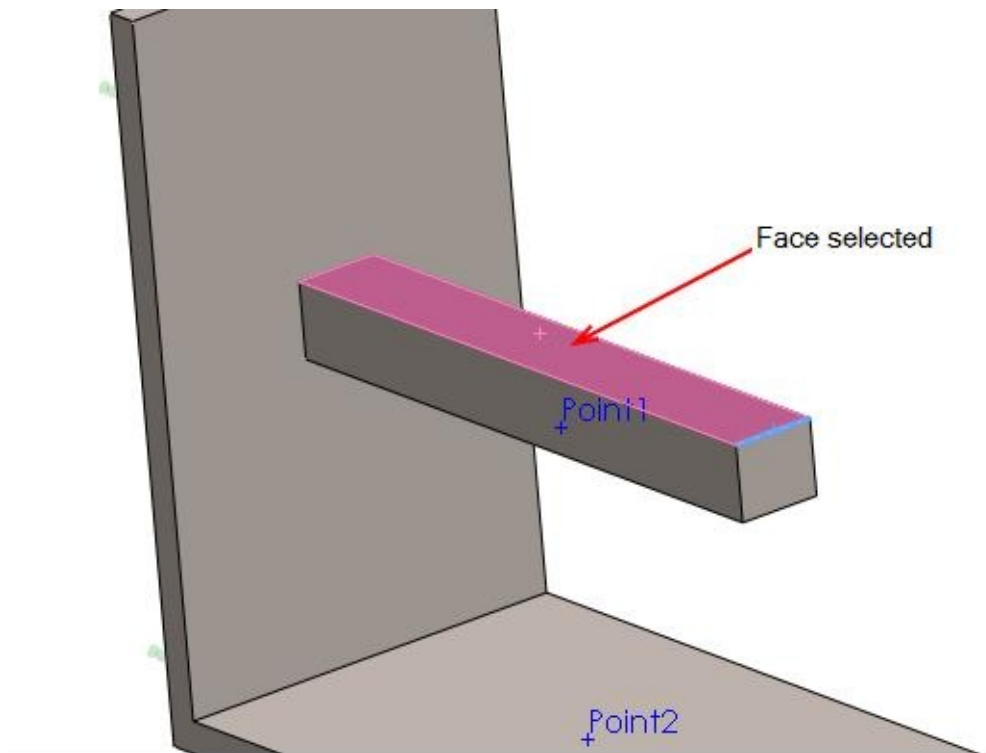


Figure-7. Face selected for force reference

- Click on the **Normal to Plane** button and specify the value as **100** N in the respective edit box.
- Click on the **OK** button from the **PropertyManager**. The force will be applied; refer to [Figure-8](#).

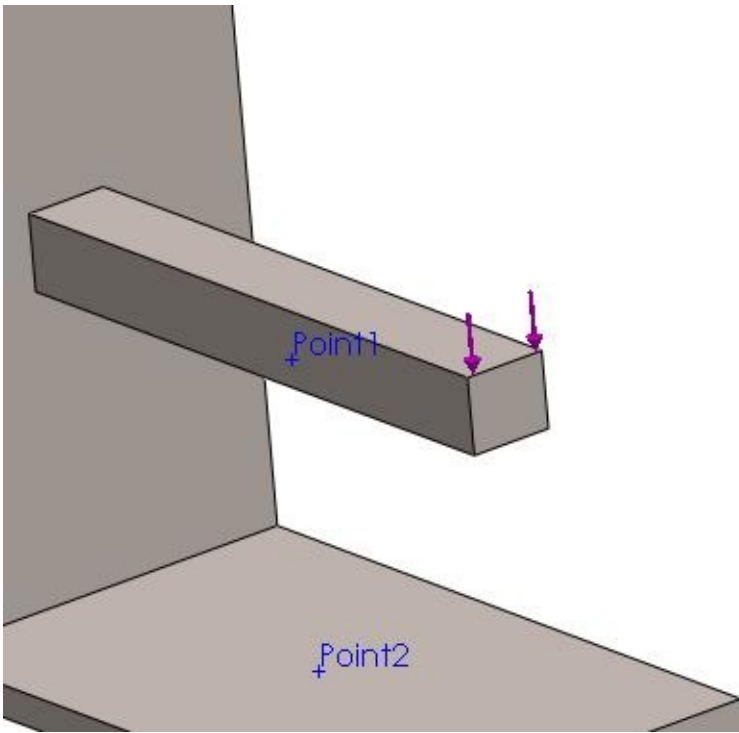


Figure-8. Force applied

Applying Connections

- Click on the down arrow below the **Connections Advisor** button and select the **Spring** button. The **Spring-Damper Connector PropertyManager** will be displayed; refer to [Figure-9](#).



Figure-9. Spring Damper Connector PropertyManager

- Select the **Compression Only** button from the **Type** rollout in the **PropertyManager**. The options in the **PropertyManager** will be changed.
- One by one select the two points displayed on the model; refer to [Figure-10](#). Make sure that you have selected the **View Points** button from the **Heads-up View Toolbar**.

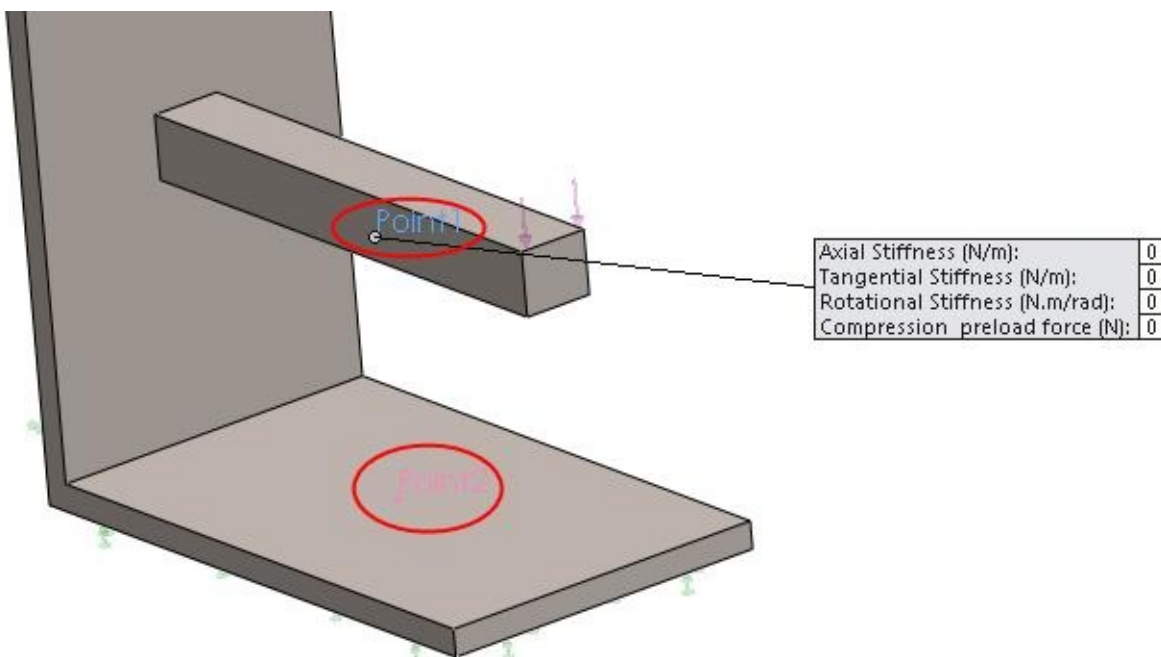


Figure-10. Points to be selected

- Enter the values in the **Options** rollout of **PropertyManager** as shown in [Figure-11](#).

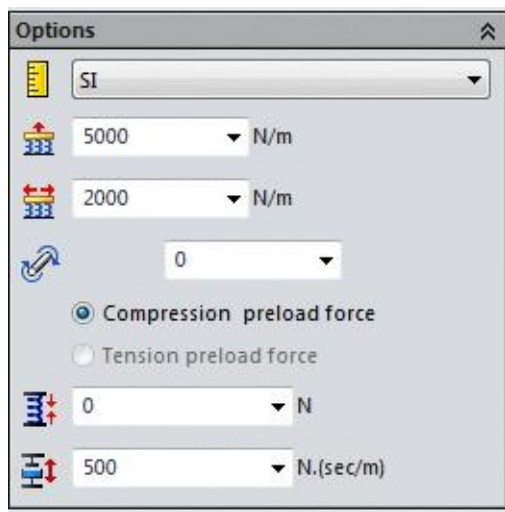


Figure-11. Options rollout

- Click on the **OK** button from the **PropertyManager**. A spring with damper will be added between the specified points; refer to [Figure-12](#).

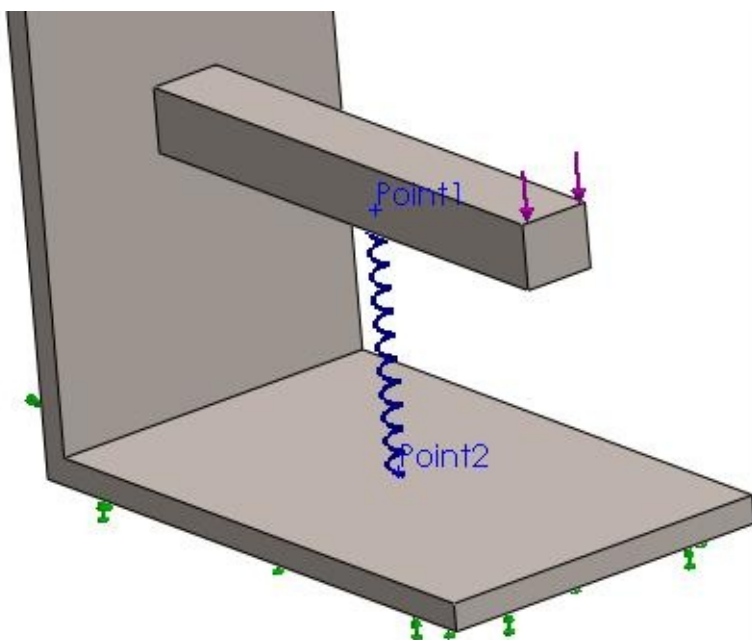


Figure-12. Spring between points

Global Damping

Global Damping option is available in all the dynamic analyses in SolidWorks Simulation since we need to specify the damping coefficient for the study. Now, assume that the system on which we are going to perform the study is submerged in oil. To let SolidWorks Simulation understand that our system is submerged in a kind of oil, we need to specify global damping matrix. The steps to do so are given next.

- Click on the **Global Damping** button from the **Ribbon**. The **Rayleigh damping PropertyManager** will be displayed; refer to [Figure-13](#).



Figure-13. Rayleigh damping PropertyManager

- Specify the damping parameters and click on the **OK** button.

Running the analysis

- Click on the **Run** button from the **Ribbon**. The system will start analyzing the problem. Once the system is done, the results will be displayed in the **Modeling** area.

You can interpret the results as discussed in earlier chapters.

PRACTICAL 1

In this practical, we will find out response curve of the model under conditions given in [Figure-14](#).

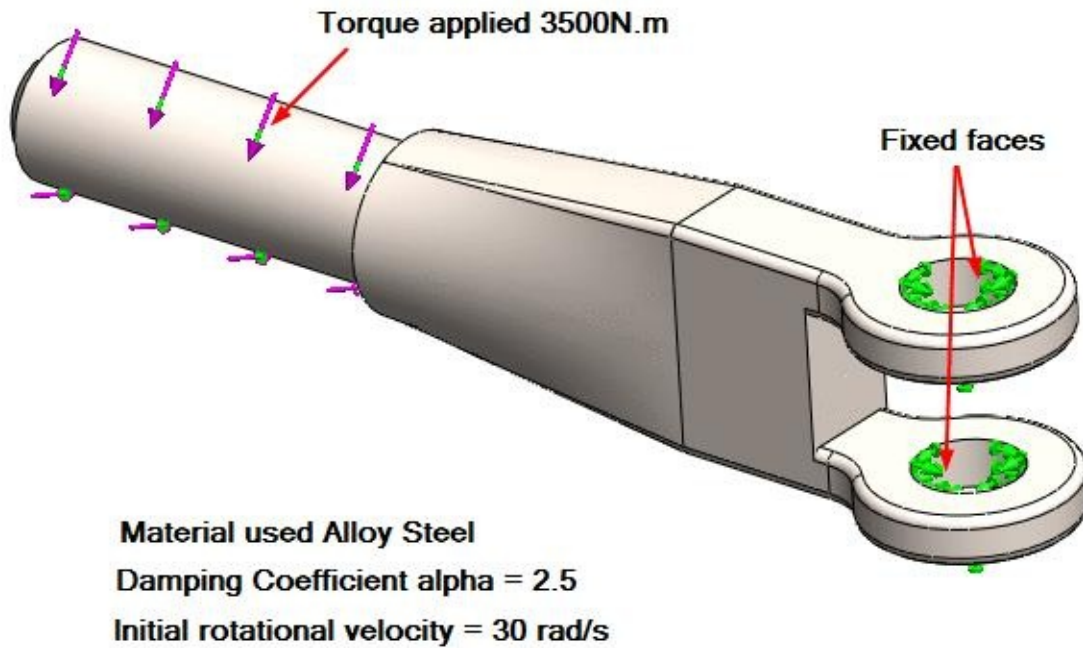


Figure-14. Practical 1 non linear dynamic

Starting Nonlinear Dynamic Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Nonlinear** button and then on the **Dynamic** button from the **Options** rollout; refer to [Figure-15](#).

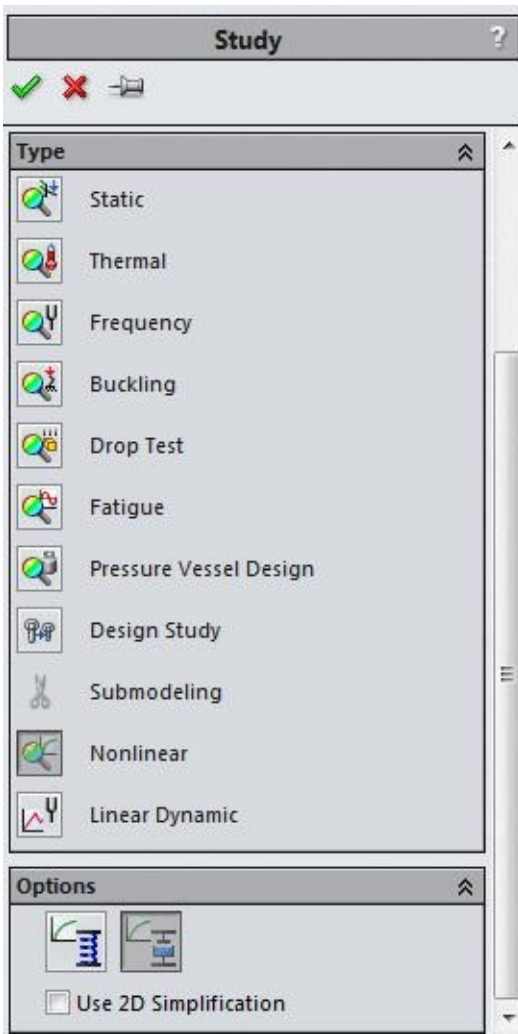


Figure-15. Study PropertyManager

- Click on the **OK** button from the **PropertyManager**. The tools related to **Non-linear Dynamic** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed as discussed in earlier chapters.
- Select the desired material, say **Alloy Steel** option from the left list and click on the **Apply** button.
- Click on the **Close** button from the dialog box to exit. The color of model will be changed; refer to [Figure-16](#).

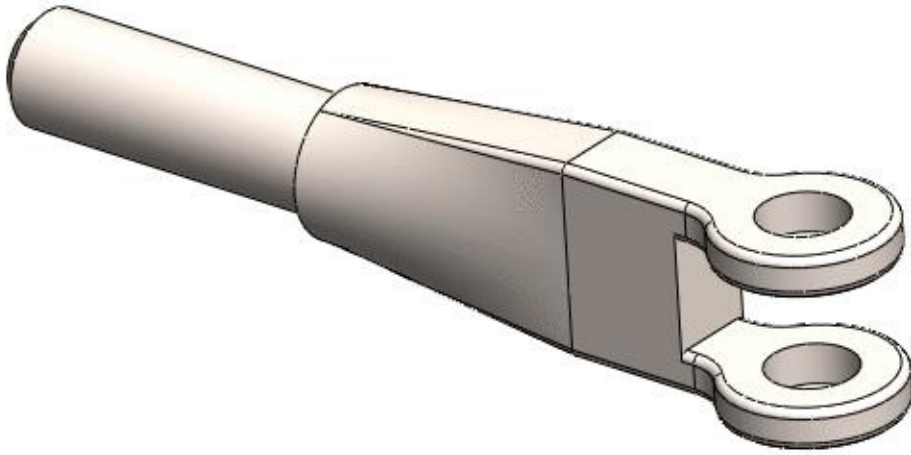


Figure-16. Model after applying material

Applying Time Varying Force

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.
- Click on the **Torque** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Select the round face of the model and specify the torque value as shown in [Figure-17](#).

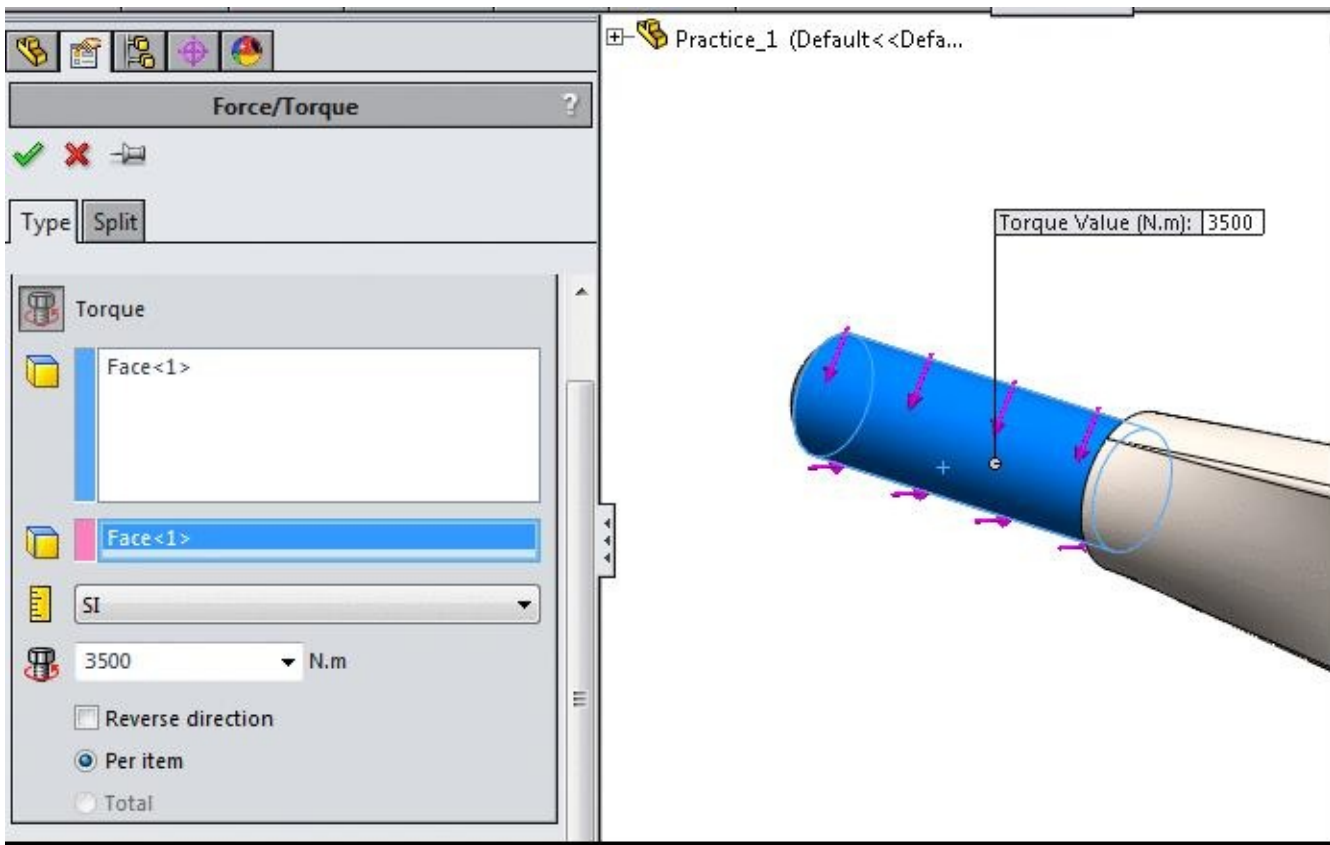


Figure-17. Torque value specified

- Expand the **Variation with Time** rollout in the **PropertyManager** and click on the **Curve** radio button to define the loading curve.
- Click on the **Edit** button from the **Variation with Time** rollout and define the time curve as shown in [Figure-18](#). Note that to add a new row in the table, you need to double-click on the last field under **Points** column in the table.

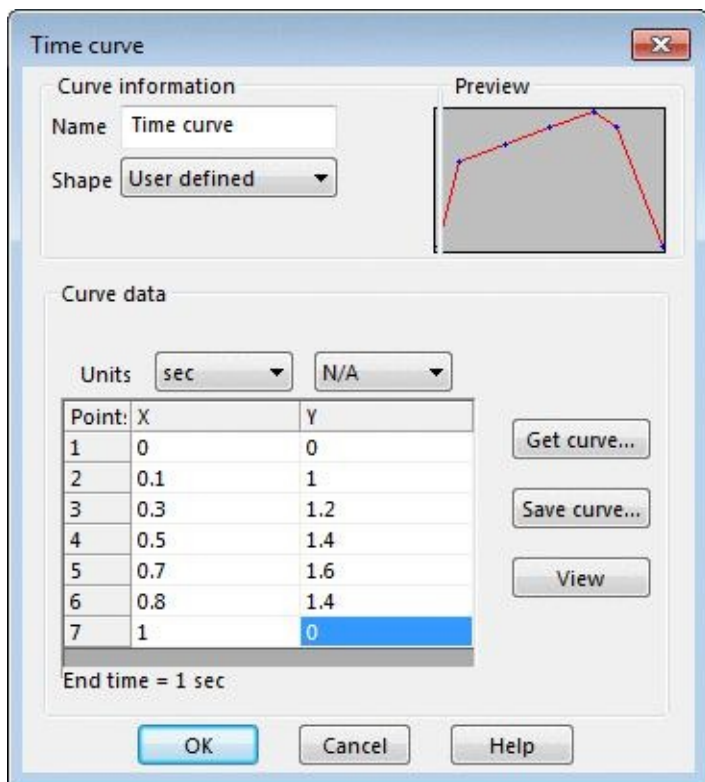


Figure-18. Time Curve for torque

- Click on the **OK** button from the dialog box and then **OK** button from the **PropertyManager**.

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed.
- Select the holes cut from the model as shown in [Figure-19](#).

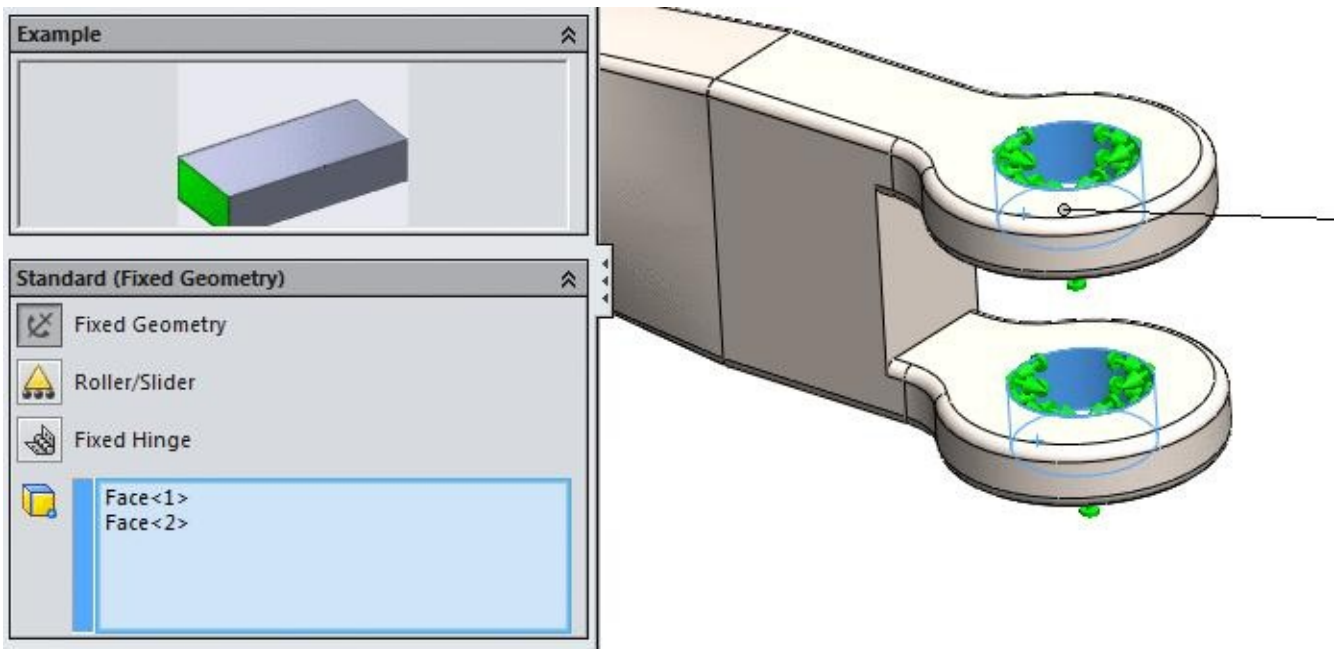


Figure-19. Holes fixed

- Click on the **OK** button from the **PropertyManager** to apply the fixtures.

Applying Damping

- Click on the **Global Damping** button from the **Ribbon**. The **Rayleigh damping PropertyManager** will be displayed.
- Specify the value of alpha as **2.5** in the corresponding edit box and then click on the **OK** button from the **PropertyManager**.

Specifying initial velocity

- Click on the **Initial Conditions** button from the **Ribbon**. The **Initial Conditions PropertyManager** will be displayed; refer to [Figure-20](#).

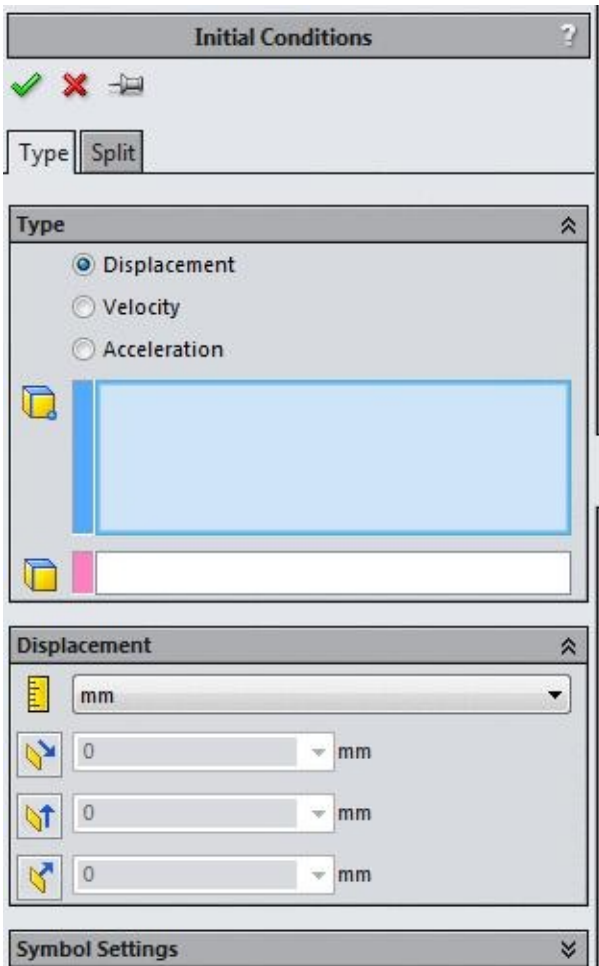


Figure-20. Initial Conditions PropertyManager

- Click on the **Velocity** radio button and select the round face of the model.
- Click in the pink selection box of **PropertyManager** to define the direction and then select the round face of the model again; refer to [Figure-21](#).

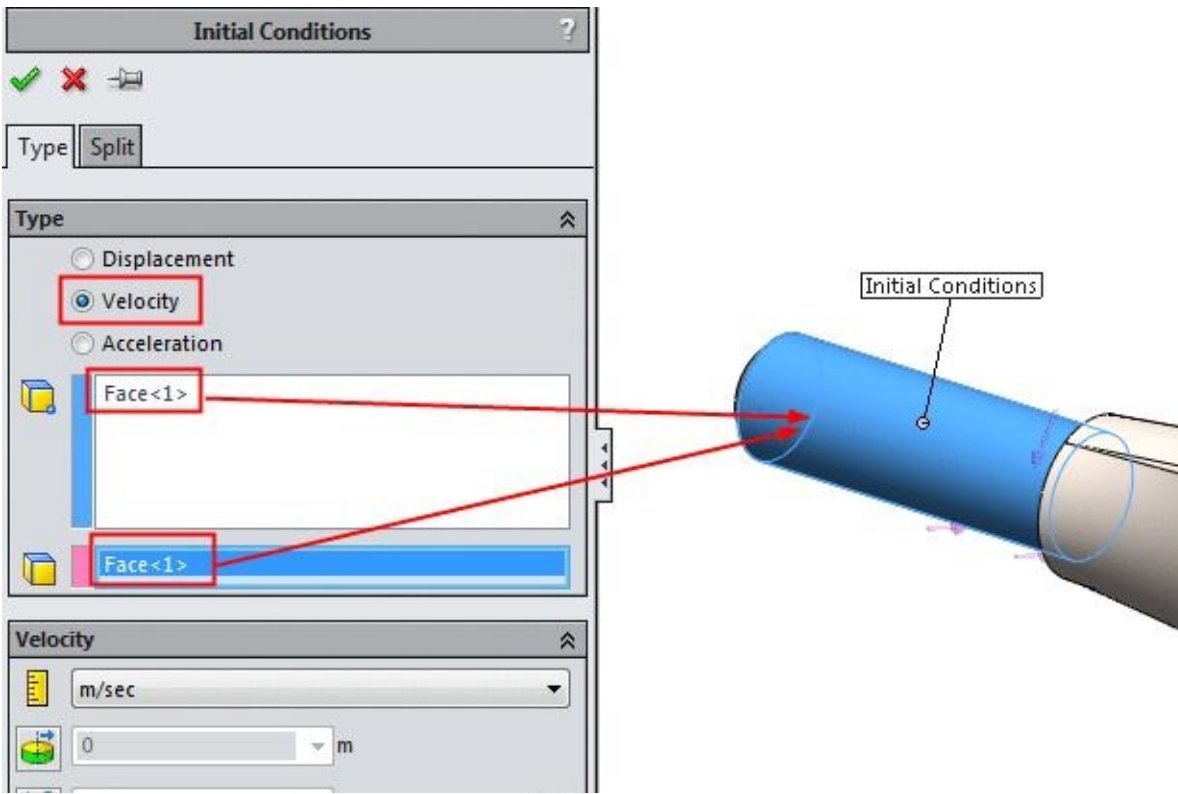


Figure-21. Face selected for defining velocity

- Select the **Circumferential** button from the **Velocity** rollout and specify the value as **30** in the corresponding edit box; refer to [Figure-22](#).

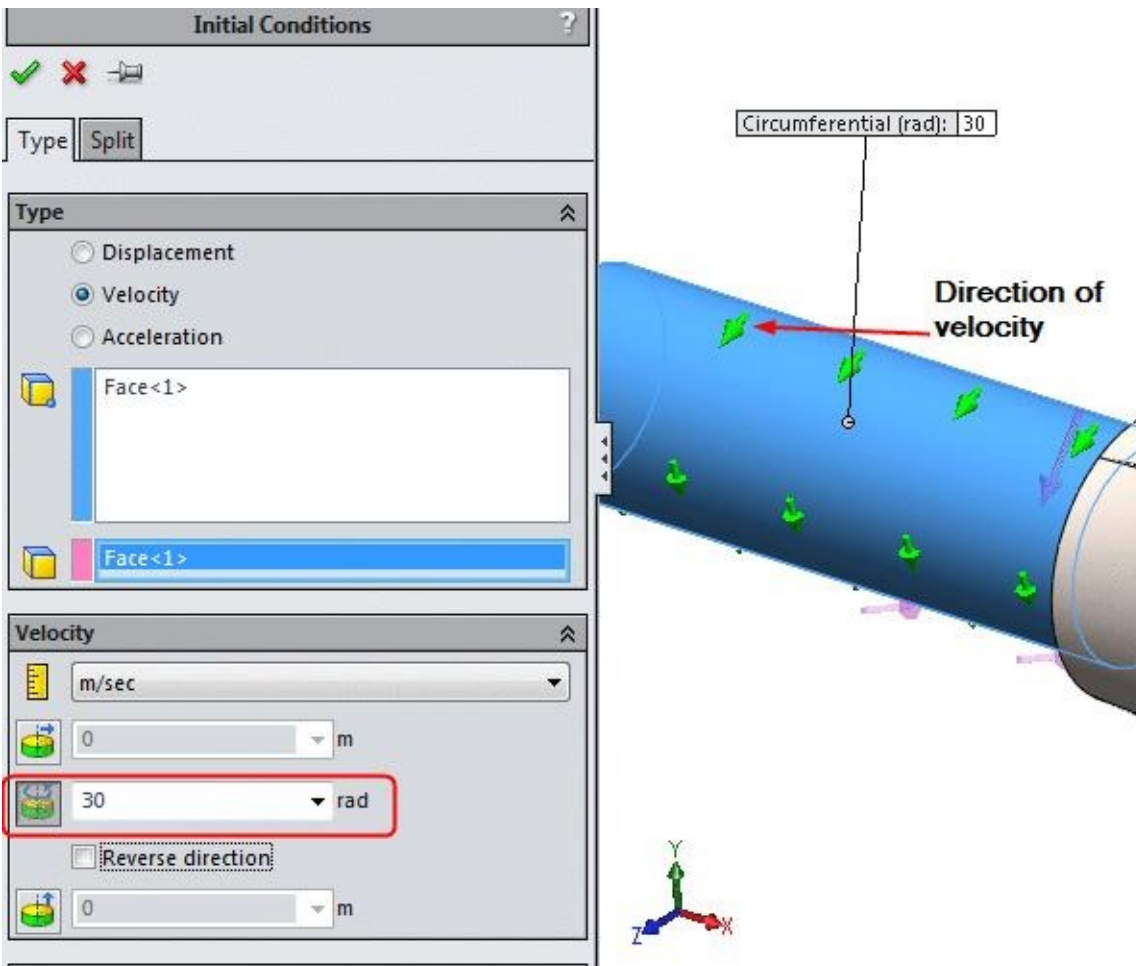


Figure-22. Circumferential velocity defined

- Click on the **OK** button from the **PropertyManager**.

Defining Mesh

Since we are performing the analysis for educational purpose only, here we will define the mesh as coarse. This will reduce the solving time of analysis.

- Click on the down arrow below **Run This Study** button in the **Ribbon**. List of tools will be displayed.
- Click on the **Create Mesh** tool from the list. The **Mesh PropertyManager** will be displayed.
- Move the slider to extreme left in the **Mesh Density** rollout in **PropertyManager**; refer to [Figure-23](#).

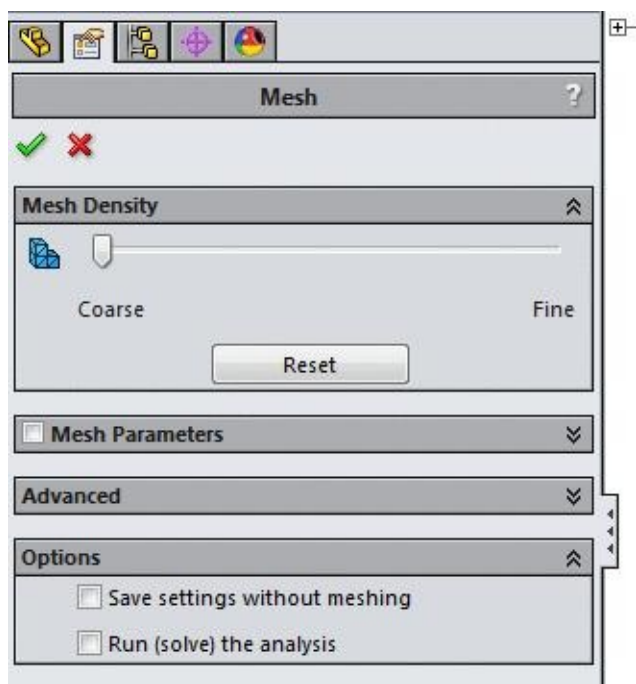


Figure-23. Coarse mesh density

- Click on the **OK** button from the **PropertyManager** to apply meshing.

Running analysis

- Click on the **Run This Study** button from the **Ribbon**. The intermediate results of analysis will be displayed; refer to [Figure-24](#). Note that the intermediate results display is latest enhancement in SolidWorks Simulation.

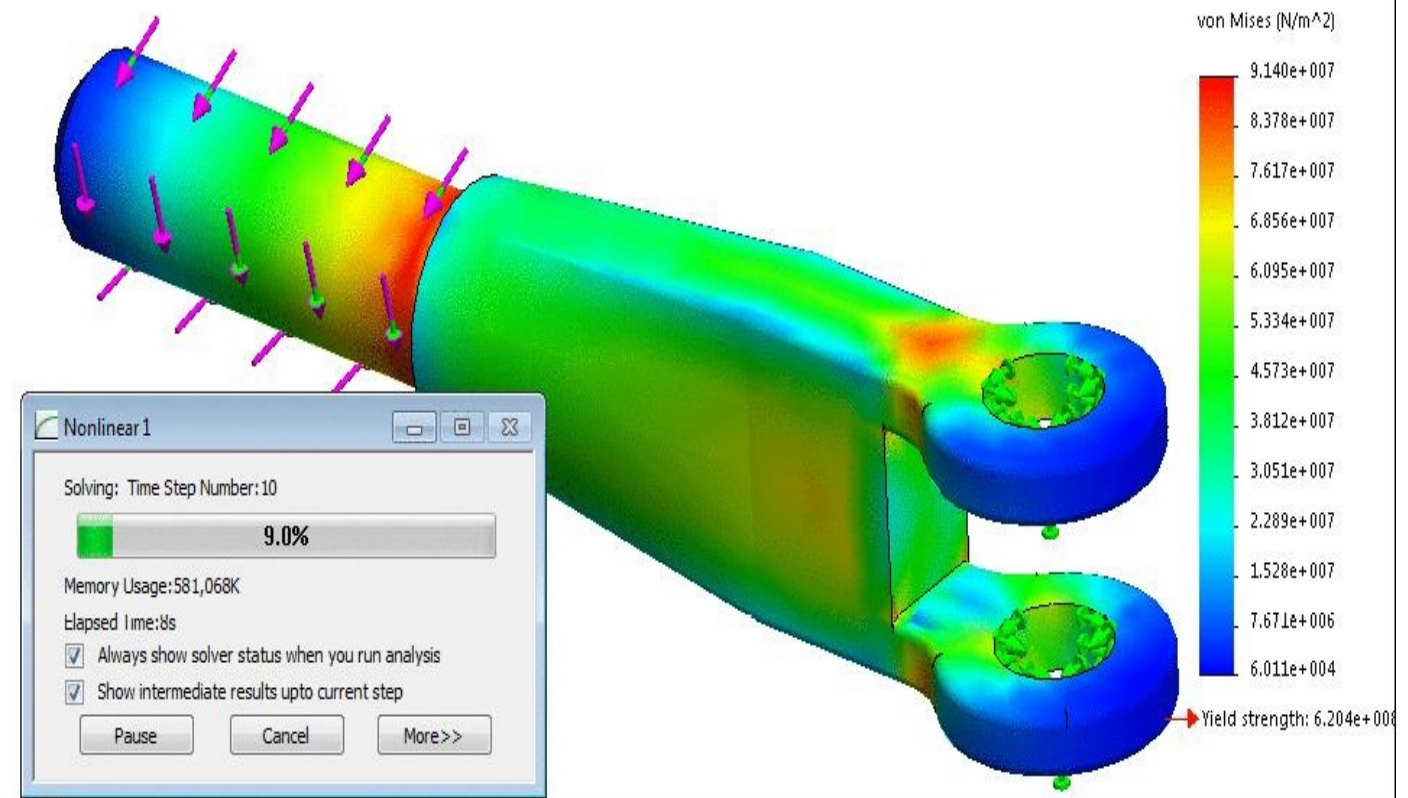


Figure-24. Running non linear analysis

Creating Time response curve

As discussed earlier, Time response curve is the main result for any Non-linear analysis because stress changes as per the force curve. It might be possible that the stress is very low at the end of study but very high in the middle of study. So, Time response curve gives the real time information of stress with respect to time. The steps to add Time response curve in results, are given next.

- Right-click on the **Result** node in the **Analysis Manager**. A short-cut menu will be displayed; refer to [Figure-25](#).

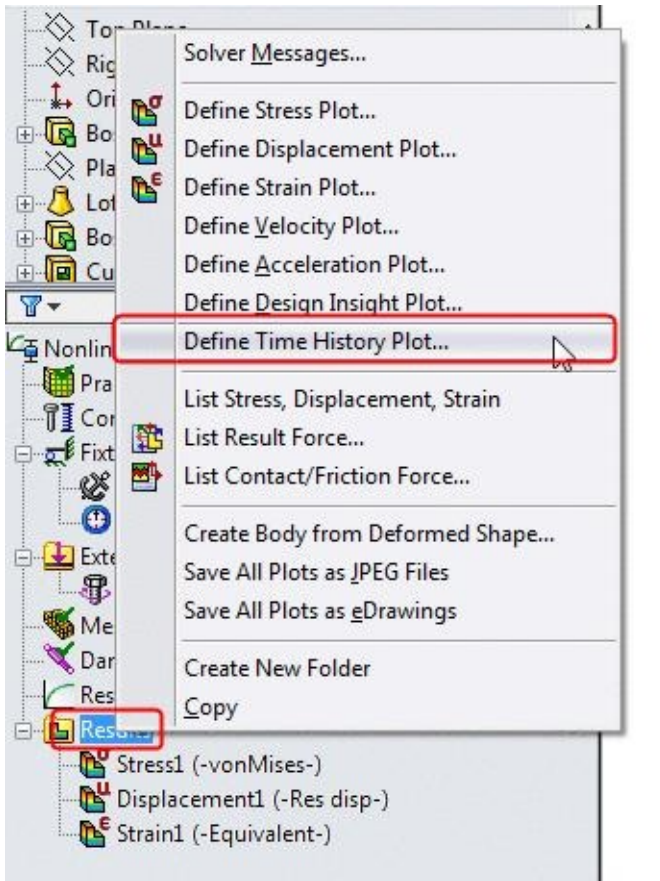


Figure-25. Shortcut menu for Time response curve

- Click on the **Define Time History Plot** option from the shortcut menu. The **Time History Graph PropertyManager** will be displayed; refer to [Figure-26](#).
- Make sure that **Time** is selected in the **X axis** drop-down and **Stress** is selected in the **Y axis** drop-down as shown in [Figure-26](#).

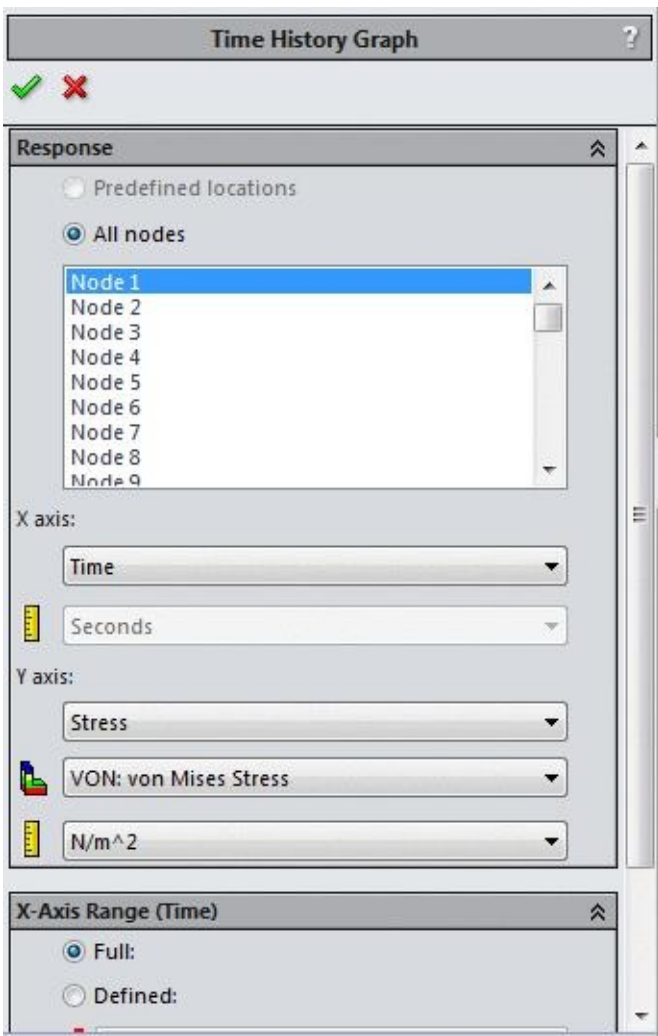


Figure-26. Time History Graph PropertyManager

- Click on the **OK** button from the **PropertyManager**. The **Response Graph** will be displayed; refer to [Figure-27](#). Note that this is the response graph of node 1 which was selected in the **Time History Graph PropertyManager** earlier; refer to previous figure ([Figure-26](#)).

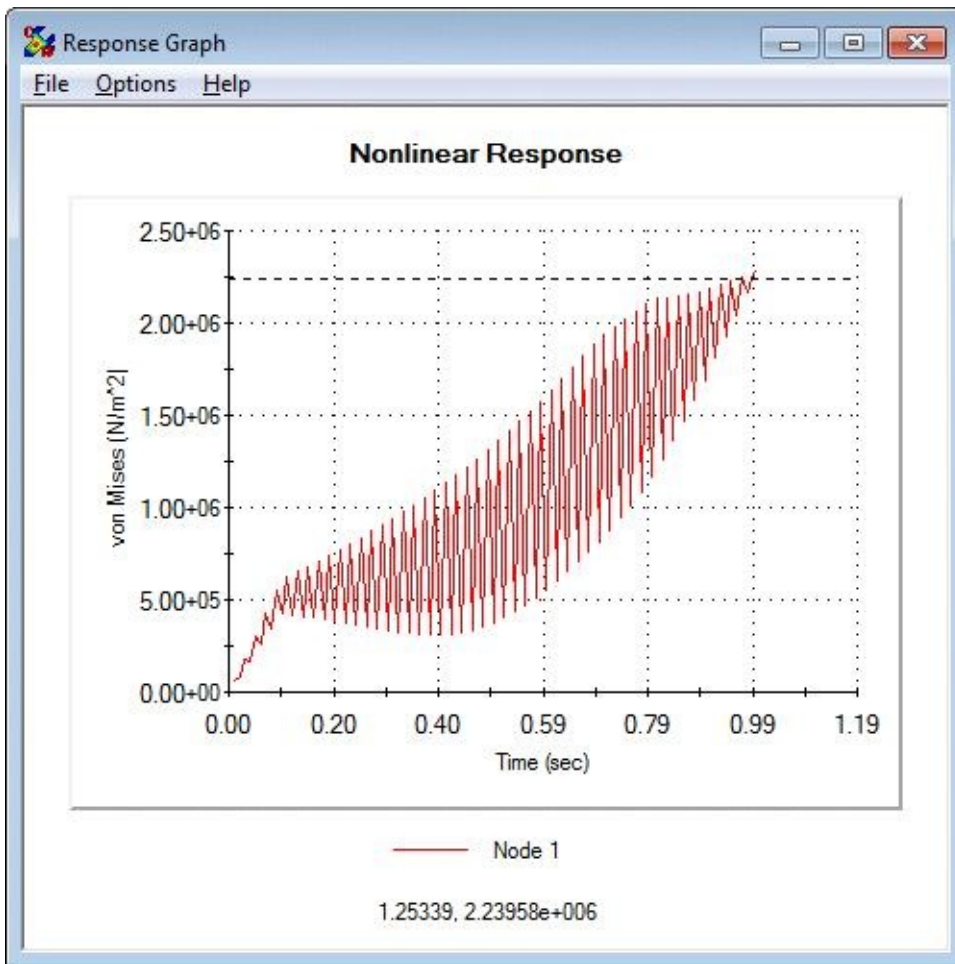


Figure-27. Response Graph

- Close the **Response Graph** dialog box and right-click on Response1 result in the **Result** node in **Analysis Manager**; refer to [Figure-28](#).

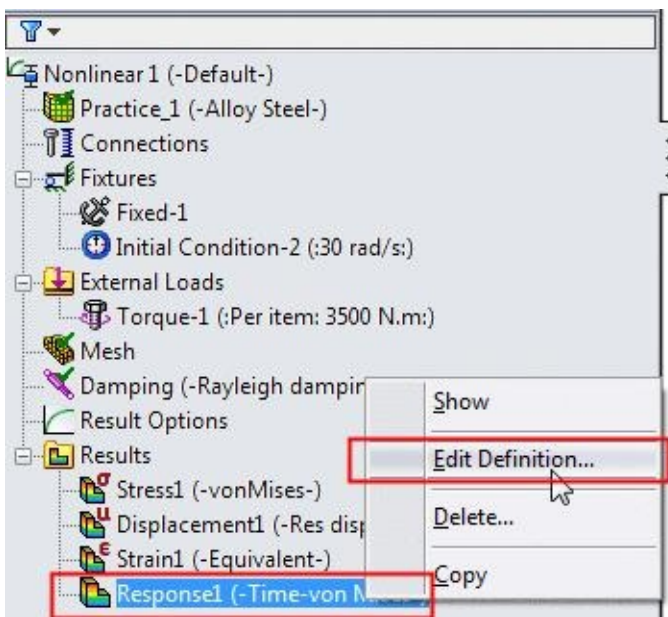


Figure-28. Shortcut menu for response result

- Select the **Edit Definition** option from the shortcut menu and select a different node to

check the response. Like, select the node 95 (Figure-29) from the list and click **OK** from the **PropertyManager** to see the response graph(Figure-30).

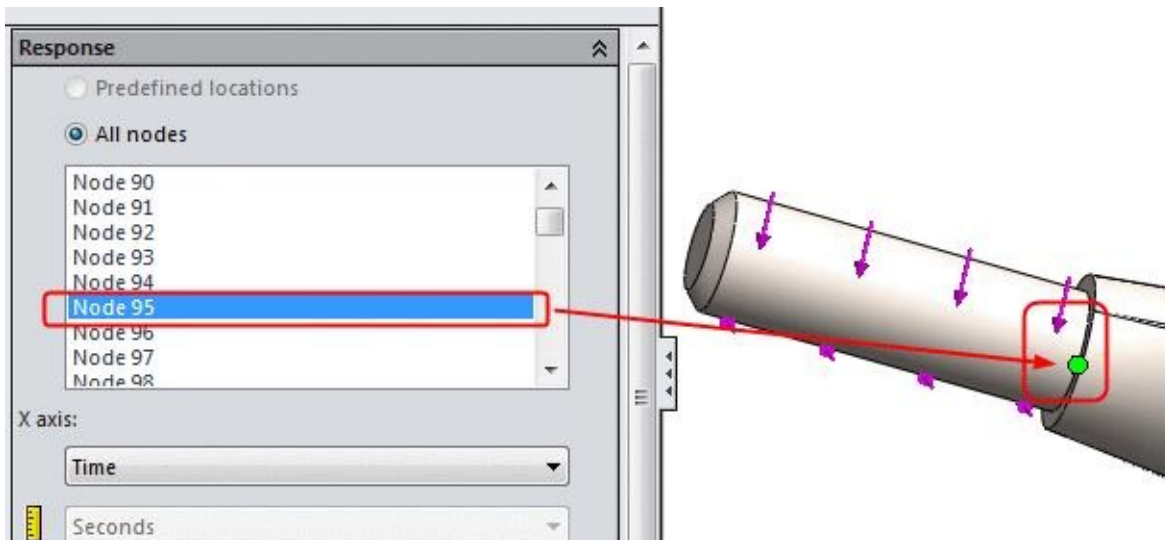


Figure-29. Node 95

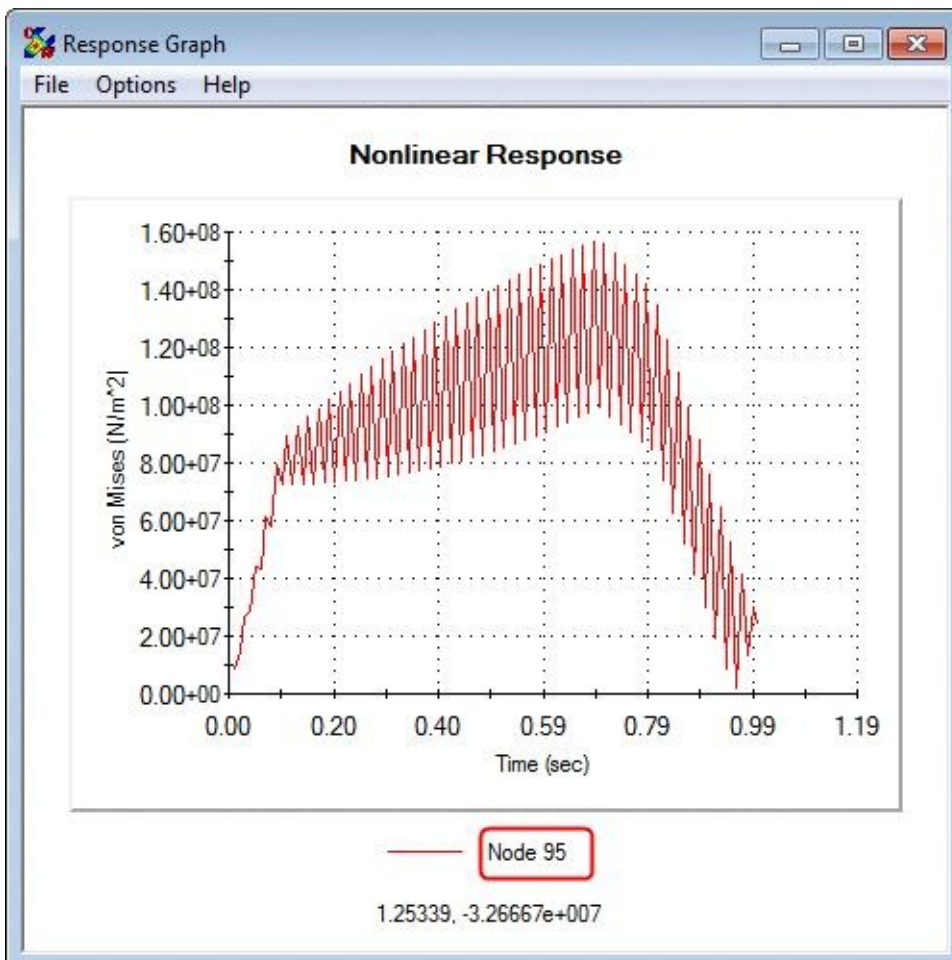


Figure-30. Response Graph for node 95

- In this way, you can check the response graph of critical areas. Note that the stress of this node has gone up to 1.566×10^8 but in Stress1 result, we don't find any spot with such high stress; refer to Figure-31. So, you can now understand why Response graph is

important in case of non-linear analyses.

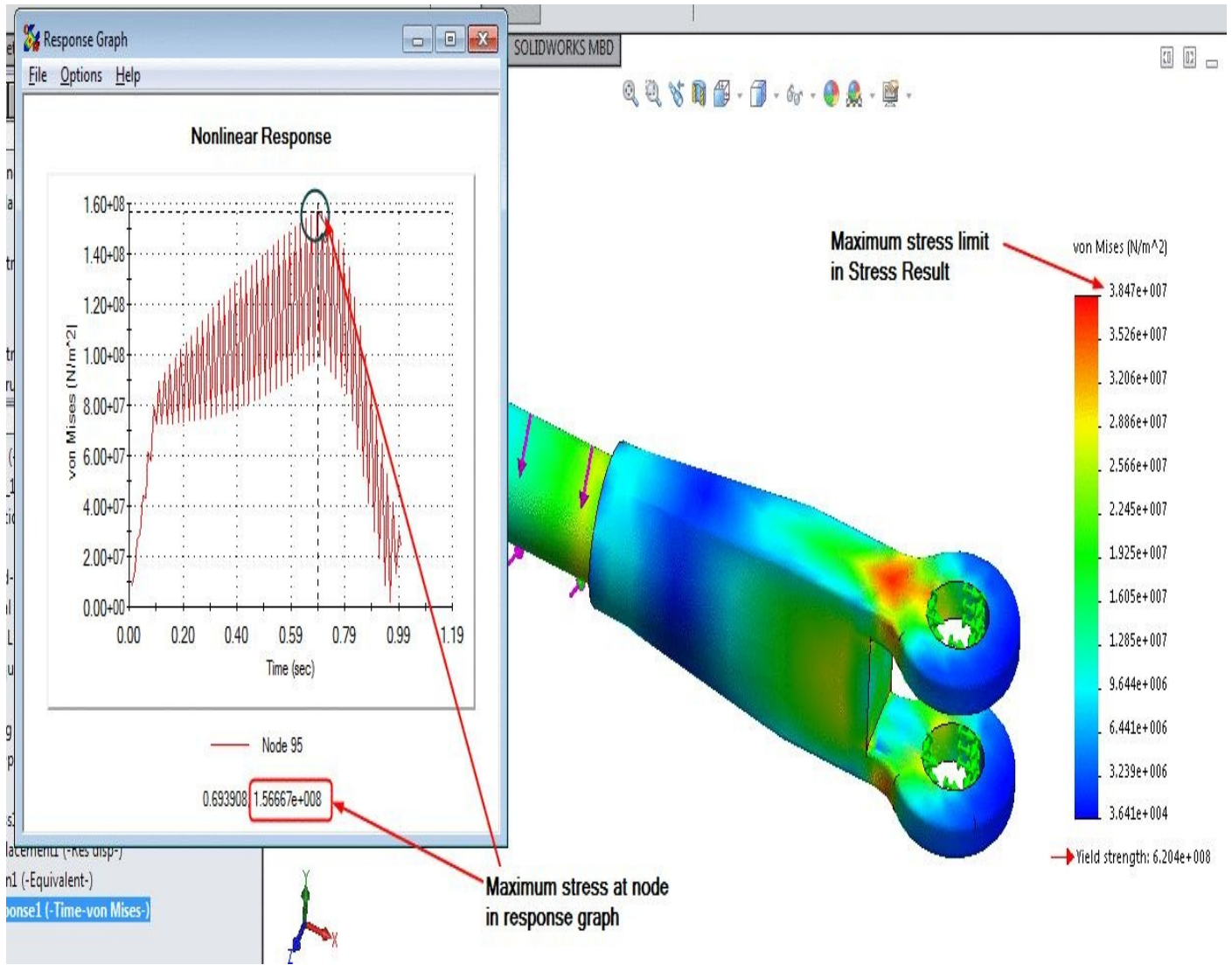


Figure-31. Comparing stress results

FREQUENCY ANALYSIS

CHAPTER 6



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Frequency Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

INTRODUCTION

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 phenomenon 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 real 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. Only the first few modes are needed for most purposes.

If your design is subjected to dynamic environments, static studies cannot be used to evaluate the response. Frequency studies can help you avoid resonance and design vibration isolation systems. They also form the basis for evaluating the response of linear dynamic systems where the response of a system to a dynamic environment is assumed to be equal to the summation of the contributions of the modes considered in the analysis.

Note that Resonance is desirable in the design of some devices. For example, resonance is required in guitars and violins.

The natural frequencies and corresponding mode shapes depend on the geometry, material properties, and support conditions. The computation of natural frequencies and mode shapes is known as modal, frequency, and normal mode 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.

Loads affect the modal characteristics of a body. In general, 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).

You do not need to define any loads for a frequency study but if you do their effect will be considered.

Note that the software also considers thermal and fluid pressure effects for frequency studies.

The procedure to perform the Frequency analysis is given next.

STARTING FREQUENCY ANALYSIS

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Frequency** button then click on the **OK** button from the **PropertyManager**. The tools related to frequency analysis will be displayed.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed.
- Expand the **SolidWorks Materials** node and then the **Aluminium Alloys** node in the left of the dialog box.
- Browse through the materials and select the **Alloy Steel** material.
- Click on the **Apply** button and then click on the **Close** button from the dialog box to exit the dialog box.

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed.
- Select the face of the model as shown in [Figure-1](#).

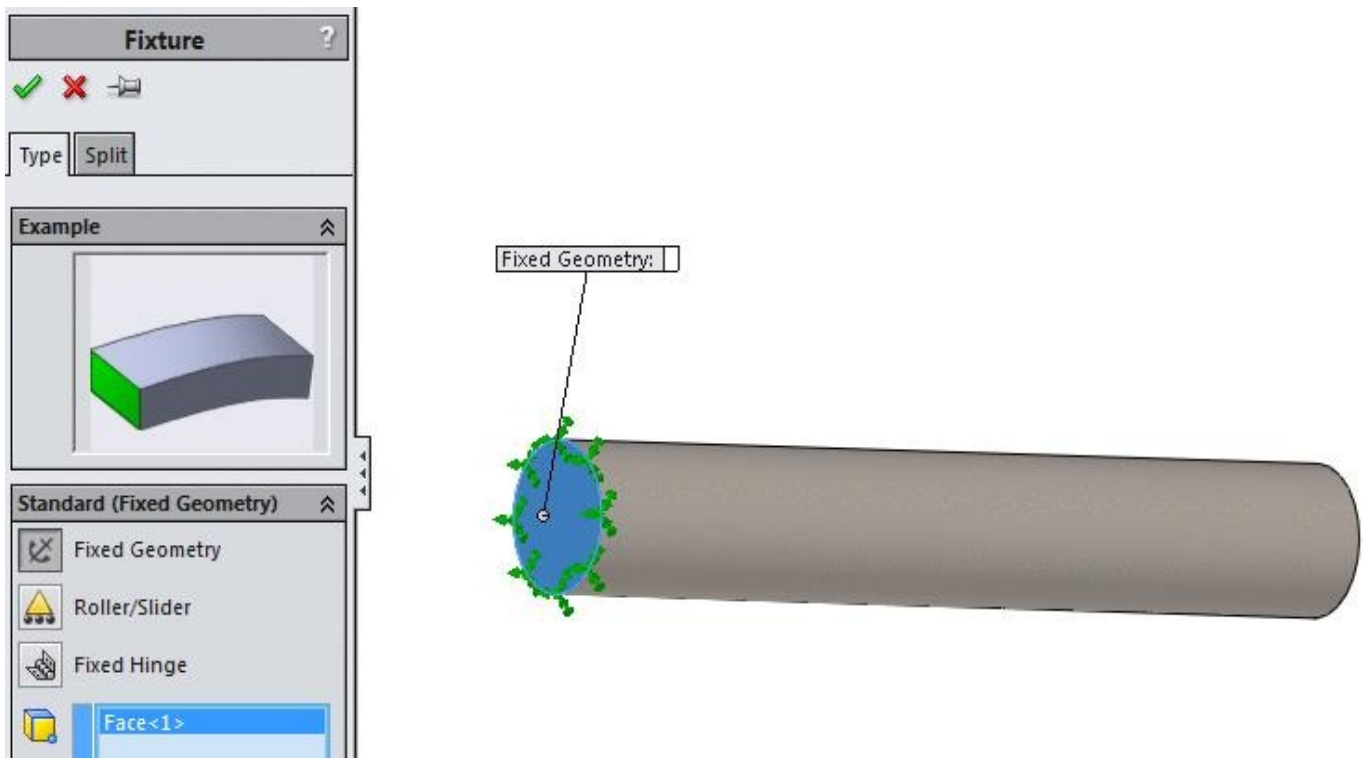


Figure-1. Face for fixture

- Click on the **OK** button from the **PropertyManager**.

Applying Forces

Note that it is not necessary to apply forces for performing the Frequency analysis but if you apply the forces, they are counted for their effects.

- Click on the down arrow below **External Loads Advisor** from the **Ribbon**. The list of tools will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed; refer to [Figure-2](#).

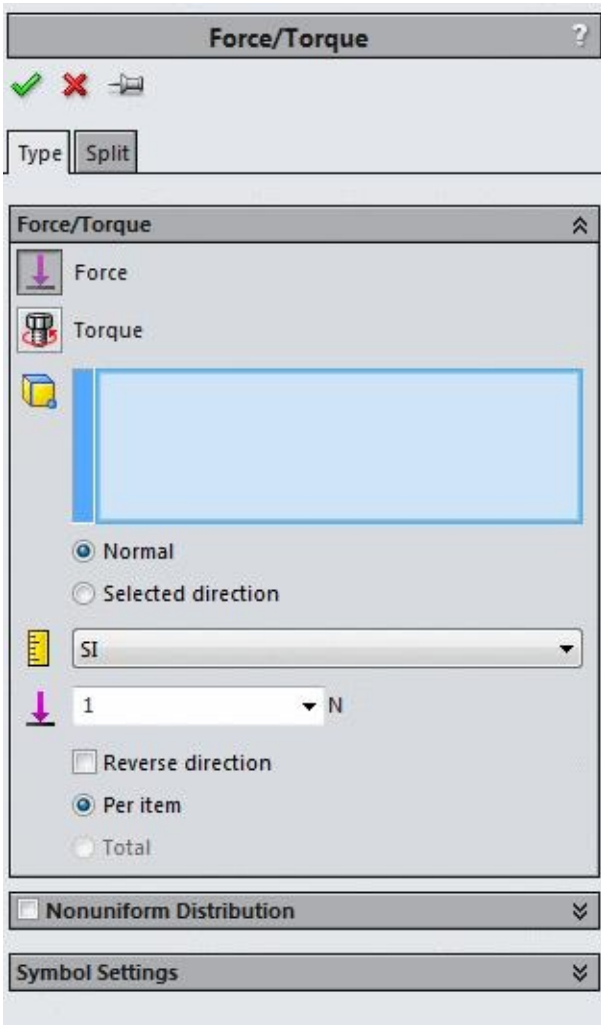


Figure-2. Force Torque PropertyManager

- Select the face as shown in [Figure-3](#). and specify the force value as 100 N.

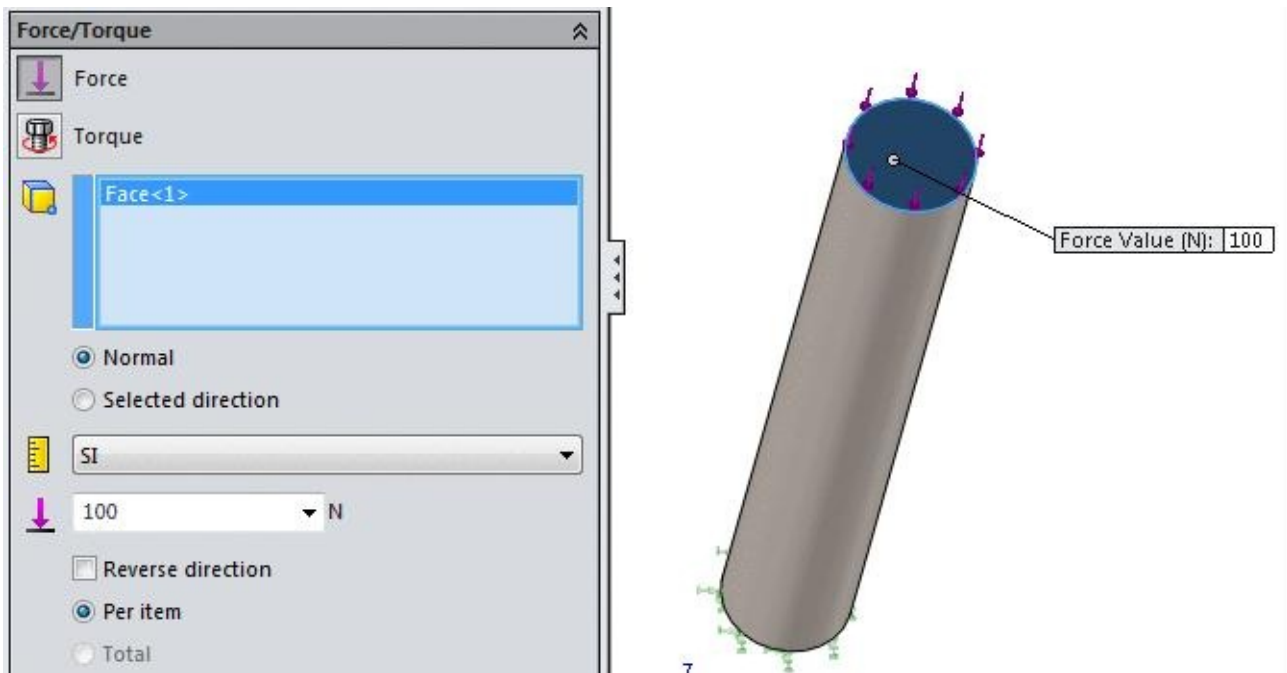


Figure-3. Force to be applied

- Click on the **OK** button from the **PropertyManager**.

Setting Parameters for the analysis

- Right-click on the name of analysis in the **Analysis Manager**. A shortcut menu will be displayed; refer to [Figure-4](#).

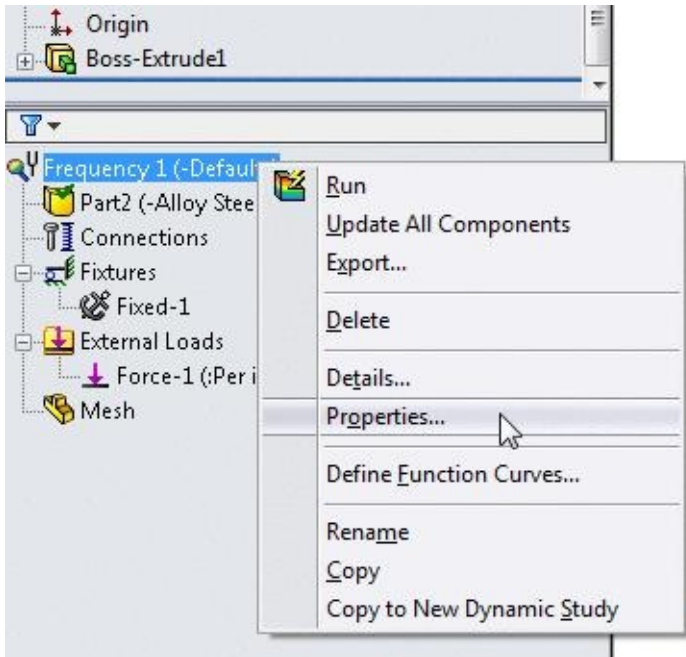


Figure-4. Shortcut menu for properties

- Click on the **Properties** button from the shortcut menu. The **Frequency** dialog box will be displayed; refer to [Figure-5](#).
- In this dialog box, the **Number of frequencies** spinner is used to specify the number of natural frequencies that are to be found out.
- If you want to check the natural frequency closer to a specific frequency then select the **Calculate frequencies closest to: (Frequency Shift)** check box and specify the value of frequency in the adjacent edit box.
- As we are applying force within the Frequency analysis, then we need to select the **Direct sparse Solver** from the **Solver** area in the dialog box. Otherwise, an error will be generated.

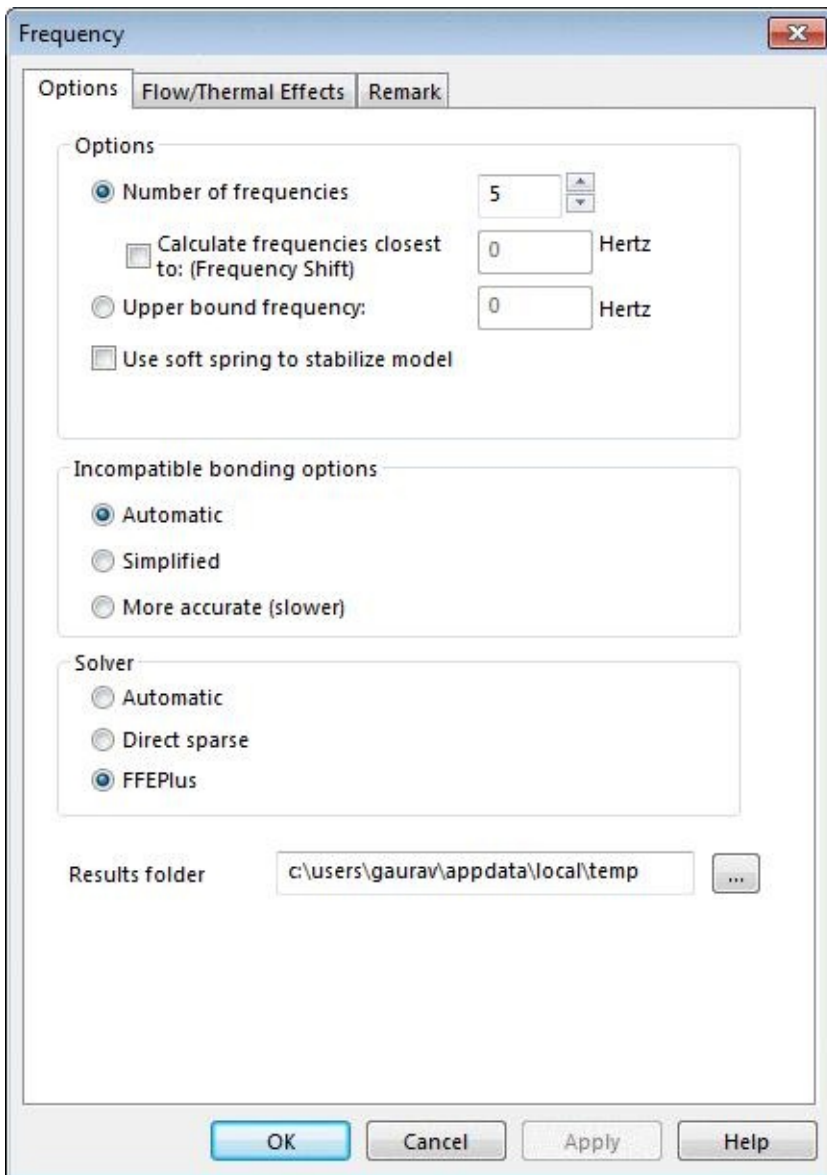


Figure-5. Frequency dialog box

- Other options in the dialog box have already been discussed.
- Click on the **OK** button from the dialog box.

Running the analysis

- Click on the **Run** button from the **Ribbon** to run the analysis. The solving process of analysis will start.
- After the process is complete, the results will be displayed; refer to [Figure-6](#).

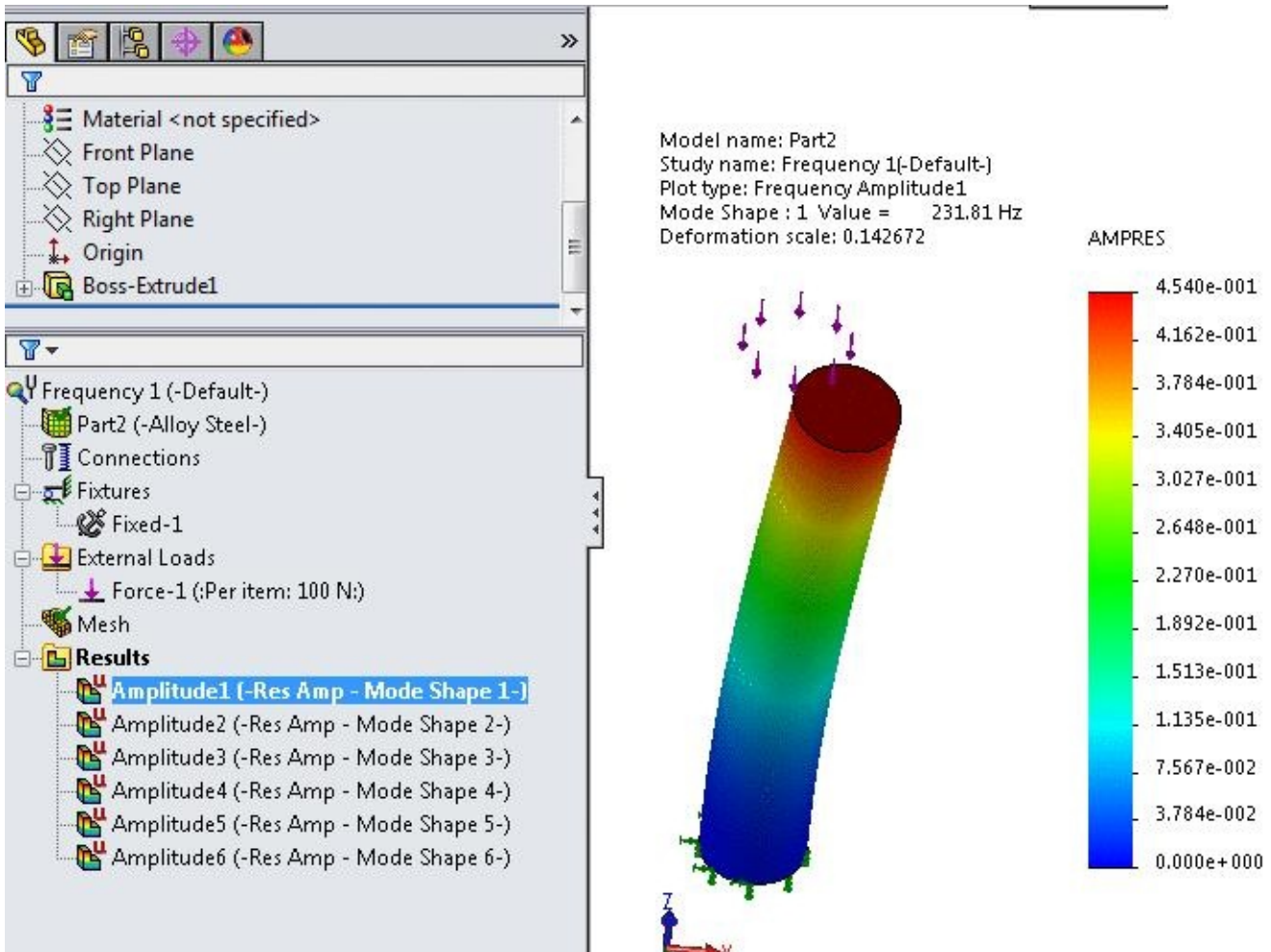


Figure-6. Result of Frequency analysis

- There are six nodes displayed under **Results** in the **Analysis Manager** which represent six mode shapes at respective natural frequencies.
- From the results, you find that our part should not be working in the range of the natural frequencies listed.
- To display the list of resonant frequencies, right-click on the **Results** node in the **Analysis Manager**. A shortcut menu will be displayed.
- Click on the **List Resonant Frequencies** button from the shortcut menu; refer to [Figure-7](#).

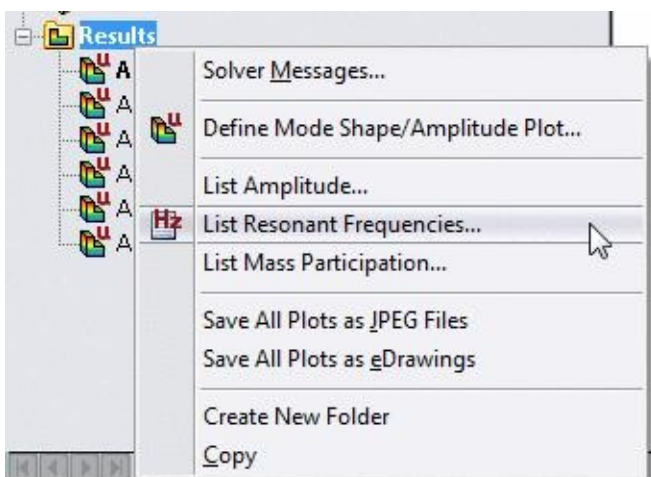
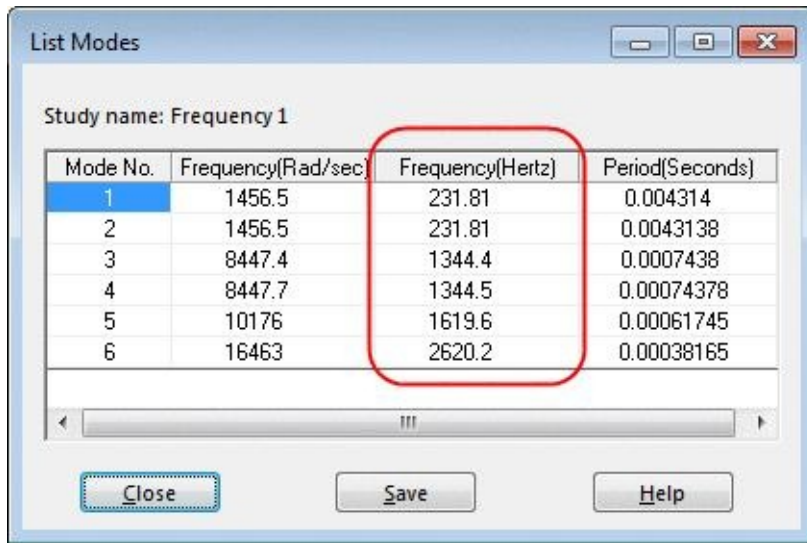


Figure-7. Shortcut menu for results

- The **List Modes** dialog box will be displayed. The **Frequency (Hz)** column shows the resonant frequencies; refer to [Figure-8](#).



The screenshot shows a dialog box titled "List Modes" with a study name of "Frequency 1". It contains a table with four columns: "Mode No.", "Frequency(Rad/sec)", "Frequency(Hertz)", and "Period(Seconds)". The "Frequency(Hertz)" column is circled in red. Below the table are three buttons: "Close", "Save", and "Help".

Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	1456.5	231.81	0.004314
2	1456.5	231.81	0.0043138
3	8447.4	1344.4	0.0007438
4	8447.7	1344.5	0.00074378
5	10176	1619.6	0.00061745
6	16463	2620.2	0.00038165

Figure-8. List Modes dialog box

LINEAR DYNAMIC ANALYSIS

CHAPTER 7



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Types of Linear Dynamic Analysis*
- *Starting Linear Dynamic Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

LINEAR DYNAMIC ANALYSIS

Static studies assume that loads are constant or applied very slowly until they reach their full values. Because of this assumption, the velocity and acceleration of each particle of the model is assumed to be zero. As a result, static studies neglect inertial and damping forces.

For many practical cases, loads are not applied slowly or they change with time or frequency. For such cases, use a dynamic study. Generally if the frequency of a load is larger than $1/3$ of the lowest (fundamental) frequency, a dynamic study should be used.

Linear dynamic studies are based on frequency studies. The software calculates the response of the model by accumulating the contribution of each mode to the loading environment. In most cases, only the lower modes contribute significantly to the response. The contribution of a mode depends on the load's frequency content, magnitude, direction, duration, and location.

Objectives of a dynamic analysis include:

- Design structural and mechanical systems to perform without failure in dynamic environments.
- Modify system's characteristics (i.e. geometry, damping mechanisms, material properties, etc.) to reduce vibration effects.

There are four type of dynamic analyses that can be performed in SolidWorks Simulation:

- Modal Time History Analysis
- Harmonic Analysis
- Random Vibration Analysis
- Response Spectrum Analysis

Before we start knowing about various dynamic analyses, it is important to understand the concept of damping.

Damping

If you apply initial conditions to a dynamic system, the system vibrates with decreasing amplitudes until it comes to rest. This phenomenon is called damping. Damping is a complex phenomenon that dissipates energy by many mechanisms like internal and

external friction, thermal effects of cyclic elastic straining materials at the microscopic level, and air resistance.

It is difficult to describe dissipation mechanisms mathematically. Damping effects are usually represented by idealized mathematical formulations. For many cases, damping effects are adequately described by equivalent viscous dampers.

A viscous damper (or dashpot) generates a force that is proportional to velocity. A piston that can move freely inside a cylinder filled with a viscous fluid like oil is an example of a viscous damper.

The following types of damping are available:

Modal Damping

Rayleigh Damping

Composite Modal Damping

Concentrated Dampers. Defined between two locations (available for modal time history analysis).

To keep our focus on SolidWorks Simulation, we are skipping the explanations of various types damping but you can find them in your course books.

On the basis of the use, the all four analyses can be explained as follows:

Modal Time History Analysis

Use modal time history analysis when the variation of each load with time is known explicitly, and you are interested in the response as a function of time.

Typical loads include:

- Shock (or pulse) loads
- General time-varying loads (periodic or non-periodic)
- Uniform base motion (displacement, velocity, or acceleration applied to all supports)
- Support motions (displacement, velocity, or acceleration applied to selected supports non-uniformly)

- Initial conditions (a finite displacement, velocity, or acceleration applied to a part or the whole model at time $t = 0$).

The solution of the equations of motion for multi degree-of-freedom systems incorporates Modal analysis techniques.

The solution's accuracy can improve by using a smaller time step.

After running the study, you can view displacements, stresses, strains, reaction forces, etc. at different time steps, or you can graph results at specified locations versus time. If no locations are specified in Result Options, results at all nodes are saved.

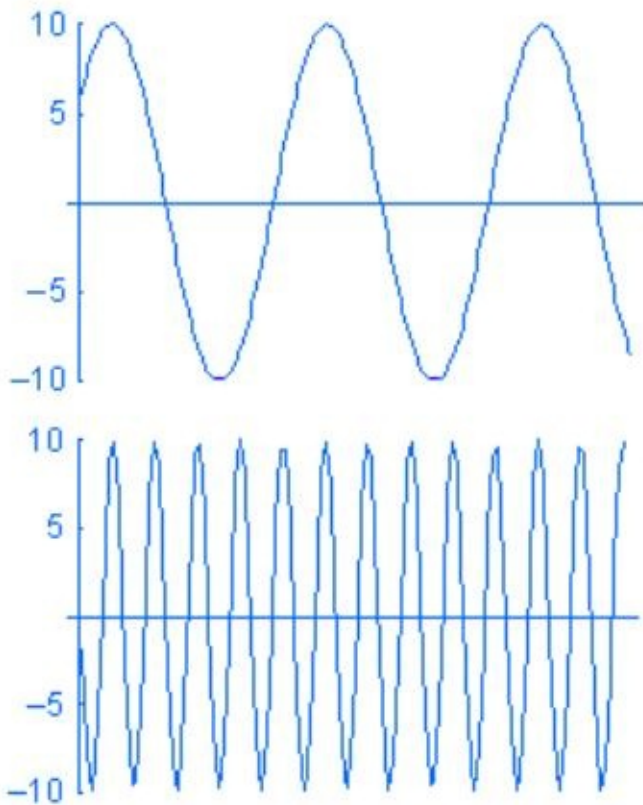
Modal, Rayleigh, composite modal, and concentrated dampers are available for modal time history analysis.

Harmonic Analysis

Use harmonic analysis to calculate the peak steady state response due to harmonic loads or base excitations.

A harmonic load P is expressed as $P = A \sin(\omega t + \phi)$ where: A is the amplitude, ω is the frequency, t is time, and ϕ is the phase angle. Sample harmonic loads of different frequencies w versus time are shown in [Figure-1](#).

Figure-1. Graph for linear dynamic analysis



Although you can create a modal time history study and define loads as functions of time, you may not be interested in the transient variation of the response with time. In such cases, you save time and resources by solving for the steady-state peak response at the desired operational frequency range using harmonic analysis.

For example, a motor mounted on a test table transfers harmonic loads to the support system through the bolts. You can model the supporting system and define a harmonic study to evaluate the steady-state peak displacements, stresses, etc. for the motor's range of operating frequencies. You can approximate the motor by a distributed mass.

After running the study, you can view peak stresses, displacements, accelerations, and velocities as well as phase angles of the response over the range of operating frequencies.

Modal, Rayleigh, and Composite modal damping options are available for this type of analysis.

Random Vibration Analysis

Use a random vibration study to calculate the response due to non-deterministic loads. Examples of non-deterministic loads include:

- Loads generated on a wheel of a car traveling on a rough road
- Base accelerations generated by earthquakes
- Pressure generated by air turbulence
- Pressure from sea waves or strong wind

In a random vibration study, loads are described statistically by power spectral density (psd) functions. The units of psd are the units of the load squared over frequency as a function of frequency. For example, the units of a psd curve for pressure are (psi)²/Hz.

The solution of random vibration problems is formulated in the frequency domain.

After running the study, you can plot root-mean-square (RMS) values, or psd results of stresses, displacements, velocities, etc. at a specific frequency or graph results at specific locations versus frequency values.

Modal, Rayleigh, and Composite modal damping options are available for this type of analysis.

Response Spectrum Analysis

In a response spectrum analysis, the results of a modal analysis are used in terms of a known spectrum to calculate displacements and stresses in the model. For each mode, a response is read from a design spectrum based on the modal frequency and a given damping ratio. All modal responses are then combined to provide an estimate of the total response of the structure.

You can use a response spectrum analysis rather than a time history analysis to estimate the response of structures to random or time-dependent loading environments such as earthquakes, wind loads, ocean wave loads, jet engine thrust or rocket motor vibrations.

After reading from the above section, we can find that the pattern of force application changes in all the analyses. Now, we will start with the Random Vibration Analysis and proceed to other Dynamic analyses.

Starting Random Vibration Analysis

- Open the part on which you want to perform the analysis.

- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Linear Dynamic** button and then on the **Random Vibration Analysis** button from the **Options** rollout; refer to [Figure-2](#).

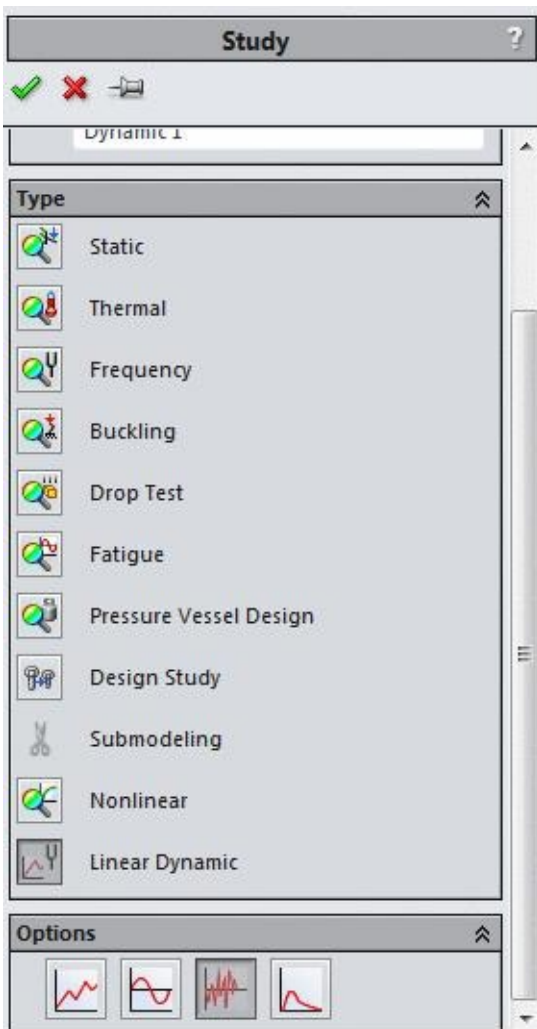


Figure-2. Study PropertyManager

- Click on the **OK** button from the **PropertyManager**. The tools related to **Linear Dynamic Random Vibration** analysis will be displayed. Note that most of the tools are same as discussed in previous chapters.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed as discussed in earlier chapters.
- Select the **Alloy Steel** option from the left list and click on the **Apply** button.
- Click on the **Close** button from the dialog box to exit. The color of model will be

changed; refer to [Figure-3](#).



Figure-3. Model after applying material

Applying Fixtures

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The tools related to fixtures will be displayed.
- Click on the **Fixed Geometry** button from the tool list. The **Fixture PropertyManager** will be displayed.
- Select the round faces of the model; refer to [Figure-4](#).

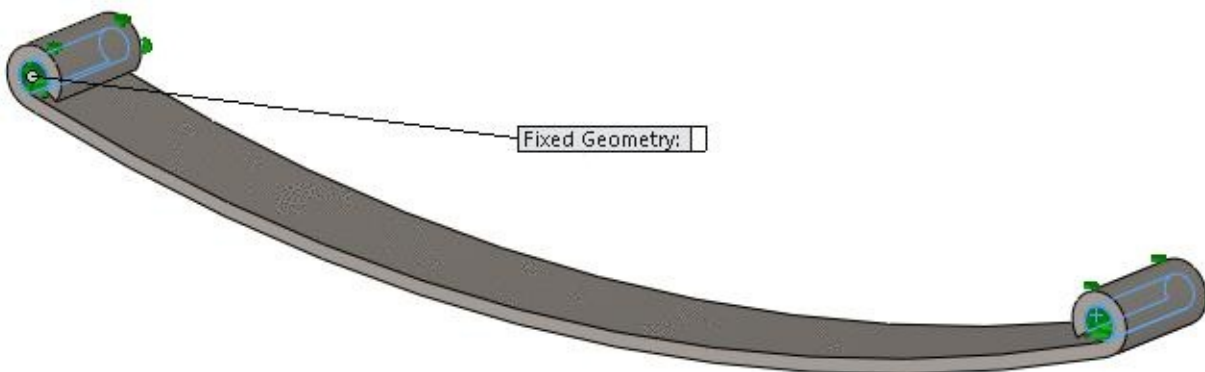


Figure-4. Faces to be fixed

- Click on the **OK** button from the **PropertyManager**. The round faces will be fixed as if they are bolted at that position.
-

Applying Force Vibrations

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of the tools related to loads will be displayed.

- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Click in the **Force Value** edit box in **Force/Torque** rollout and specify the value as **5000**.
- Select the face of the model as shown in [Figure-5](#).

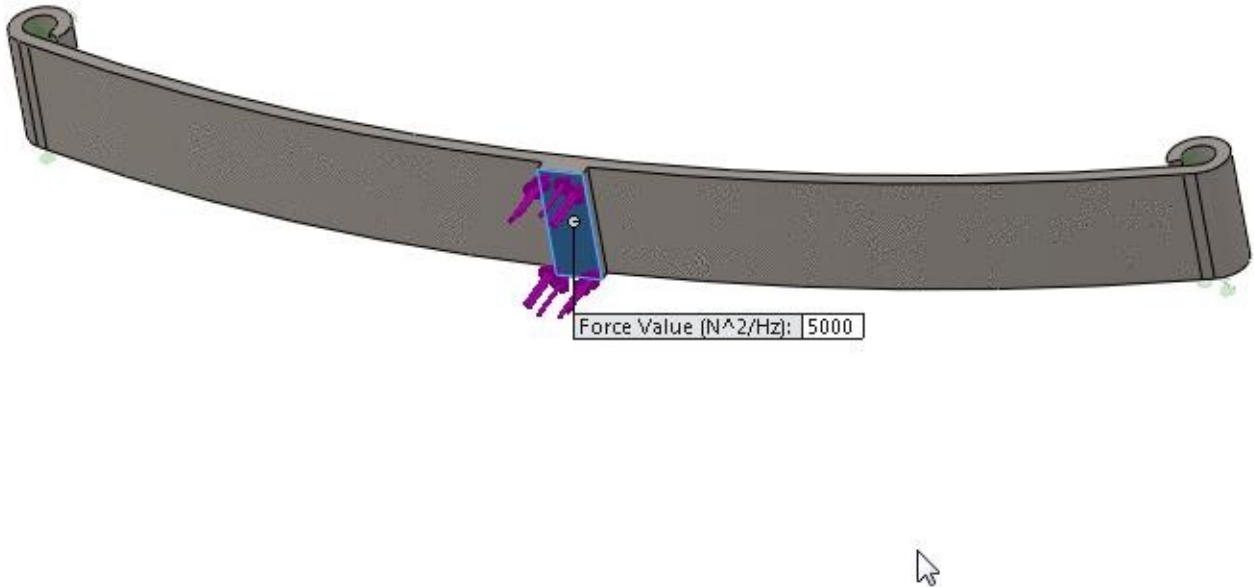


Figure-5. Face of model to be selected

- Click on the **OK** button from the **PropertyManager**.

Applying Global Damping

- Click on the **Global Damping** button in the **Ribbon**. The **Global Damping PropertyManager** will be displayed; refer to [Figure-6](#).

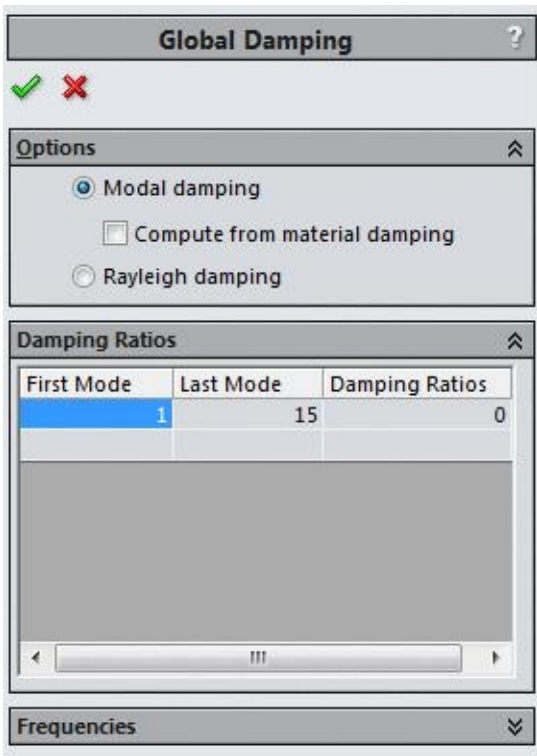


Figure-6. Global Damping PropertyManager

- Specify the modal damping ratio in the cell under **Damping Ratios** column in the table otherwise, click on the **Rayleigh damping** radio button and specify the desired parameters in the edit boxes displayed.
- Click on the **OK** button from the **PropertyManager**.

Running the analysis

- Click on the **Run** button from the **Ribbon**. The system will start analyzing the problem. Once the system is done, the results will be displayed in the **Modeling** area; refer to [Figure-7](#).

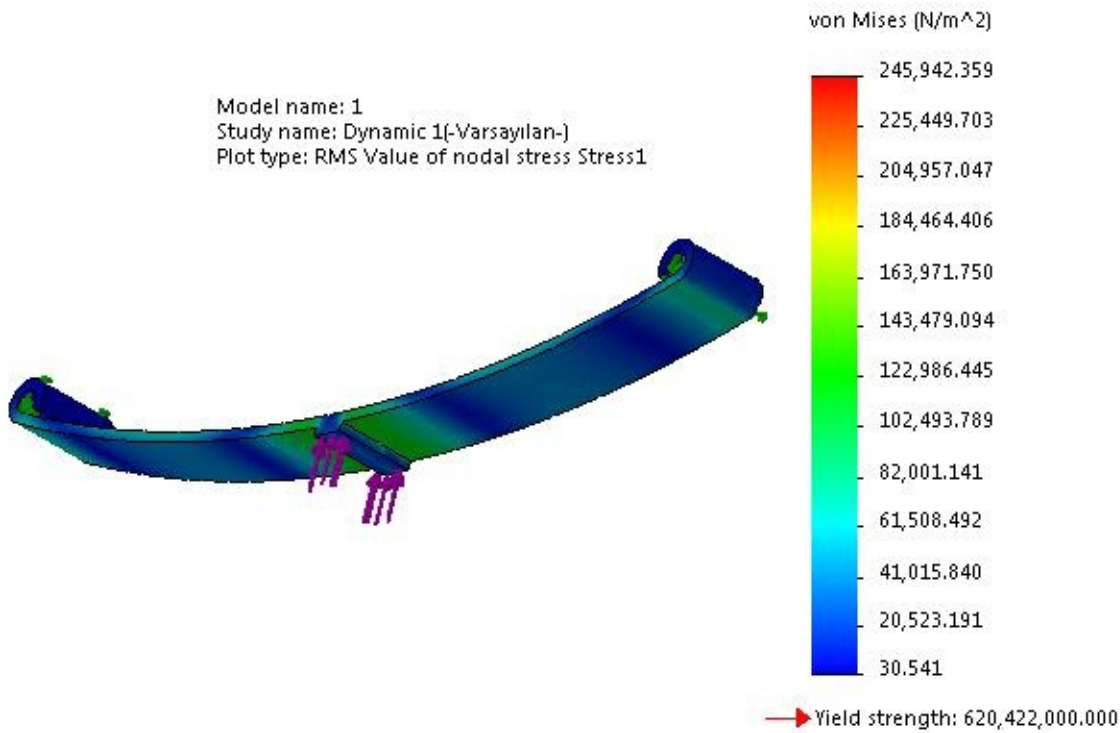


Figure-7. Result

- You may need to deselect the **Deformed Result** button from the **Ribbon** to check the non-deformed result; refer to [Figure-8](#).

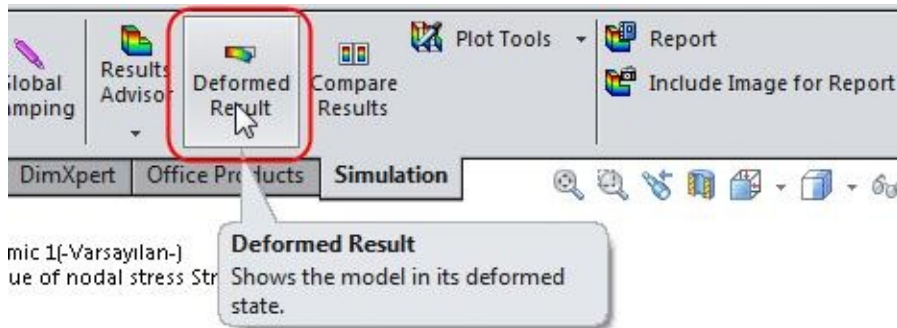


Figure-8. Deformed Result button

You can interpret the results as discussed in earlier chapters.

MODAL TIME HISTORY ANALYSIS

As discussed earlier, Modal Time History analysis is done when you know the variation of load with respect to time explicitly. Note that Time has good relation with Frequency which is $\text{Frequency} = 1/\text{Time}$. So, we can define the changing load with respect to time or Frequency. The procedure to perform this analysis is given next.

Assume that there is a bolt mounted on engine chamber. Now, you know that engine has little bit teasing nature when it is started. So, the force exerted on bolt is varying with time. The complete parameters of study are given in [Figure-9](#). We need to find out the Stress induced in the bolt.

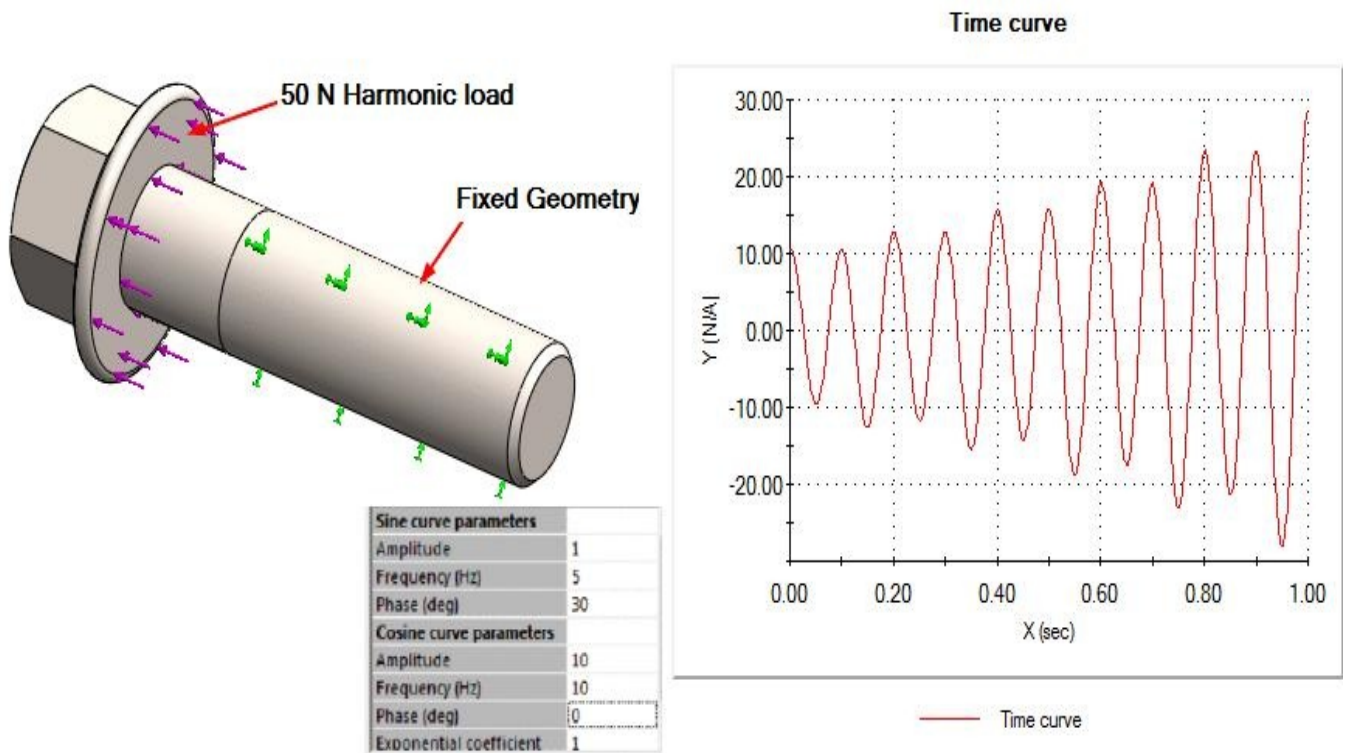


Figure-9. Parameters for Modal Time History study

Starting Modal Time History Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Linear Dynamic** button and then on the **Modal Time History** button from the **Options** rollout; refer to [Figure-10](#).

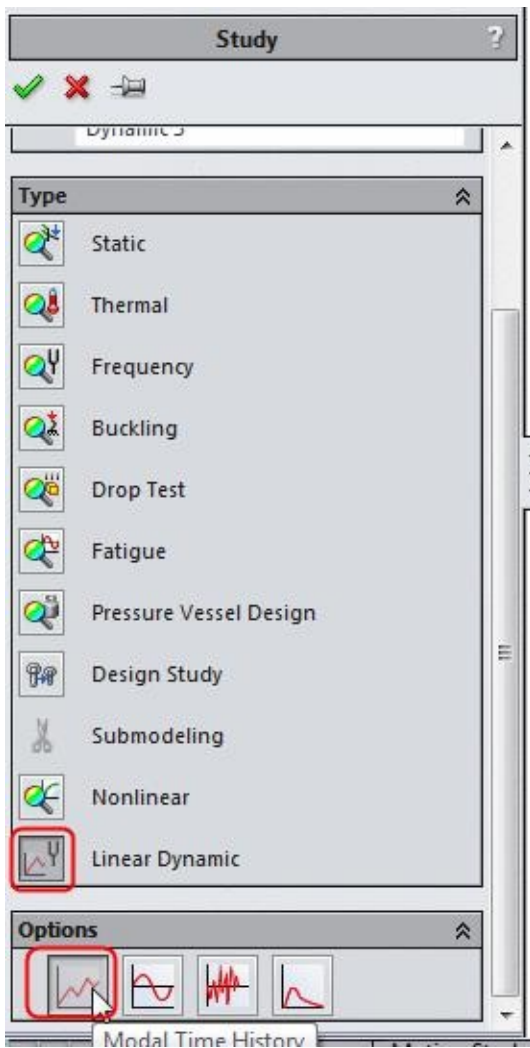


Figure-10. Modal Time History button

- Click on the **OK** button from the **PropertyManager**. The options related to Modal Time History Analysis will be displayed.

Applying Material and Fixture

- Click on the **Apply Material** button from the **Ribbon** and select the **Alloy Steel** material from the **Material** dialog box displayed.
- Click on the **Apply** button and then **Close** button from the **Material** dialog box.
- Click on the down arrow below **Fixtures Advisor** button in the **Ribbon** and select the **Fixed Geometry** button. The **Fixture PropertyManager** will be displayed.
- Select the round bottom face of the bolt as shown in [Figure-11](#).

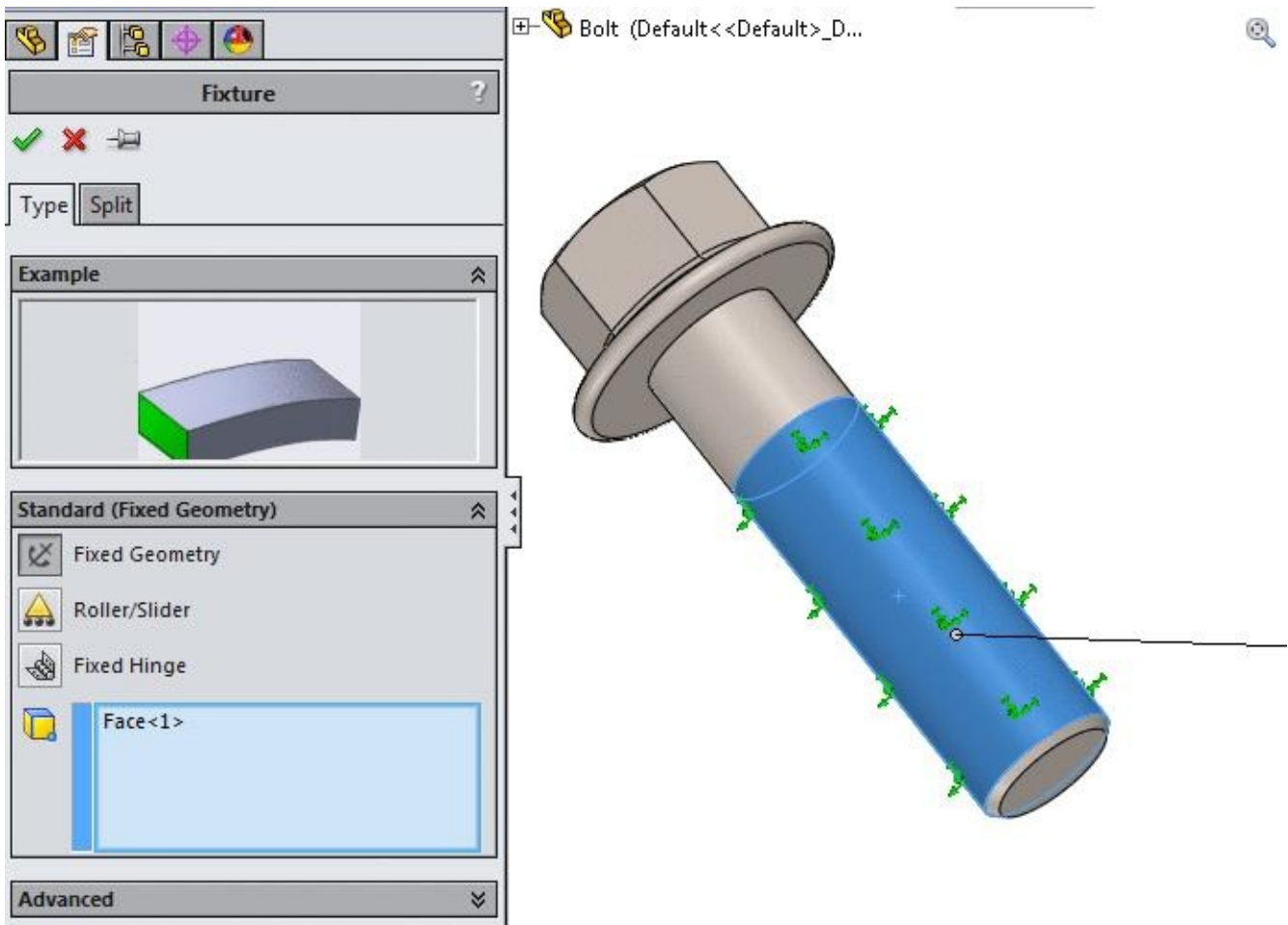


Figure-11. Fixed round face

- Click on the **OK** button from the **PropertyManager** to apply fixture.

Applying Harmonic Force

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of tools will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Select the flat face of the bolt as shown in [Figure-12](#).

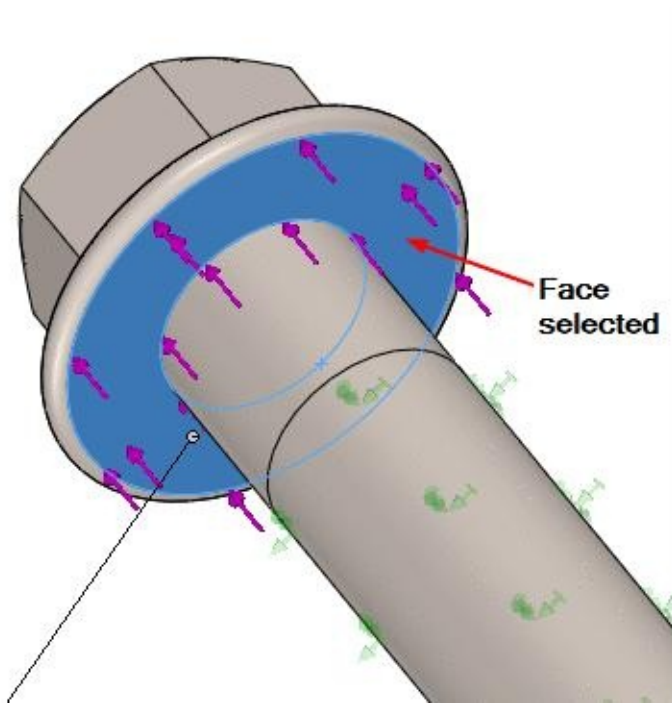


Figure-12. Face selected for applying force

- Click in the **Force Value** edit box in the **PropertyManager** and specify the value as **50 N**.
- Select the **Curve** radio button from the **Variation with Time** rollout in the **PropertyManager** and click on the **Edit** button. The **Time Curve** dialog box will be displayed.
- Select **Harmonic Loading** option from the **Shape** drop-down and specify the parameters as shown in [Figure-13](#). Specify the **Start time** and **End time** as **0** and **1** respectively, in the dialog box.

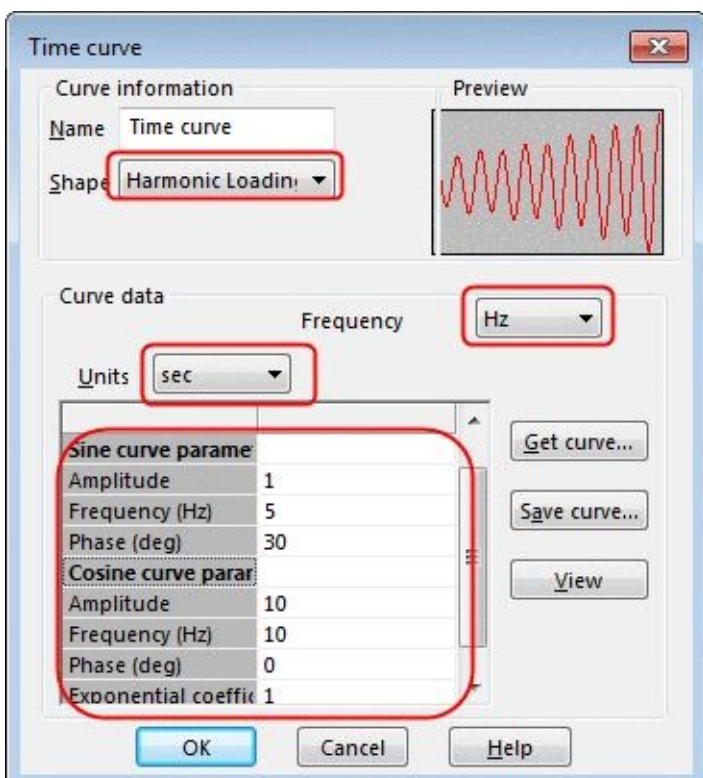


Figure-13. Parameters for Time curve

- Click on the **OK** button from the dialog box to apply the Harmonic curve.
- Click on the **OK** button from the **PropertyManager** to apply the force.

Running the Analysis

Click on the **Run This Study** button from the **Ribbon** to see the magic. Stress result will be displayed as shown in [Figure-14](#). You already know the rest of story about generating other reports like Modal Shape Plot, Frequency Response Plot, and so on.

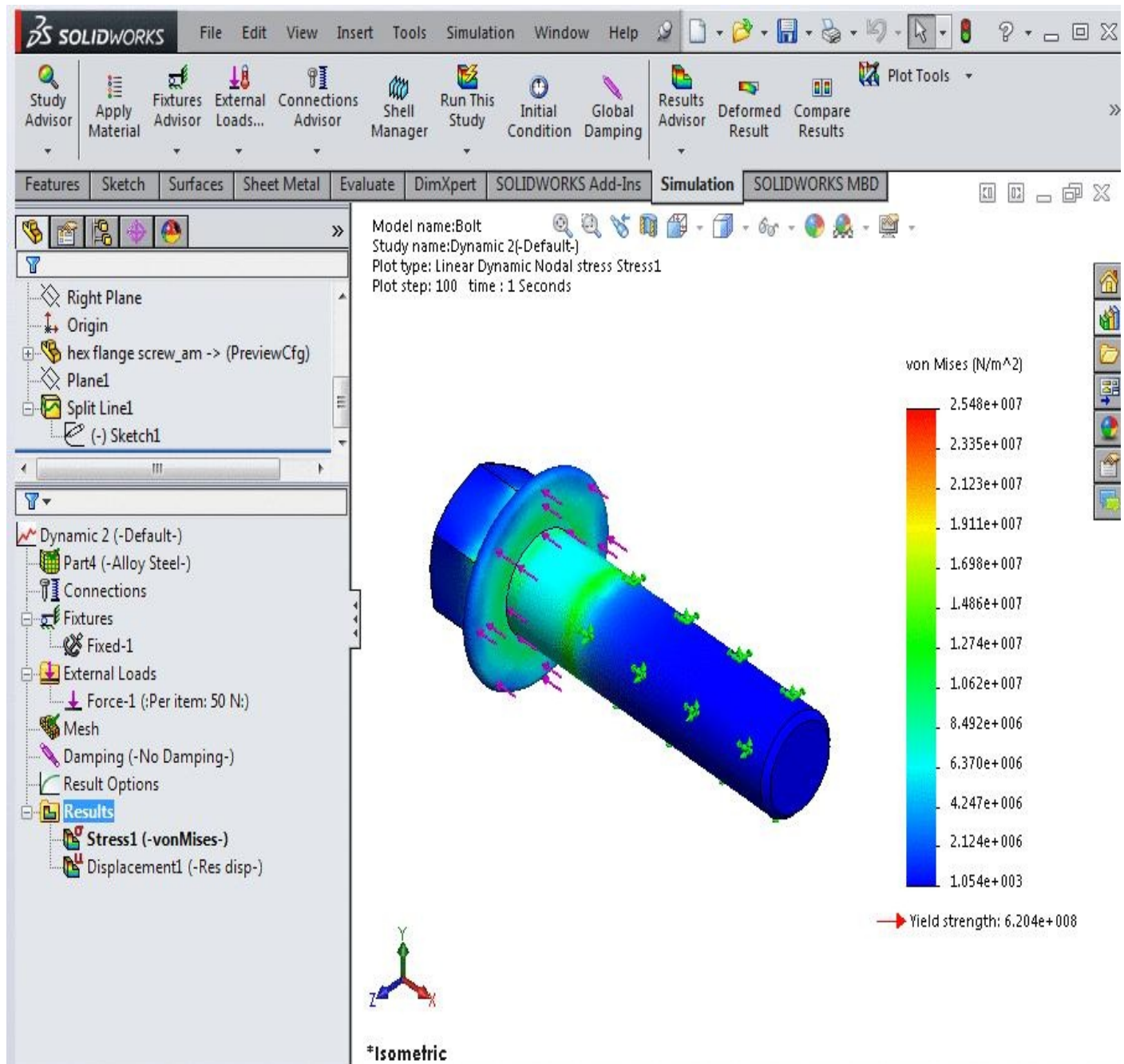


Figure-14. Stress report

In the above analysis, we have seen the effect of vibration on the bolt during starting of engine. We find that the stress induced is well below the Yield Strength. But, the journey of bolt has not ended here. Now, the bolt has to sustain the continuous harmonic loading due to vibrations of engine because Jimmy is going for New York with his bike.

HARMONIC ANALYSIS

Now, we have a range of frequency under which the bolt is going to work during his course to New York, say **0 Hz** to **5000 Hz**. Note that there might be natural frequencies of bolt in this range which can be dangerous for Jimmy's safety. Let's see whether Jimmy will reach New York or not.

Starting Harmonic Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Linear Dynamic** button and then on the **Harmonic** button from the **Options** rollout; refer to [Figure-15](#).

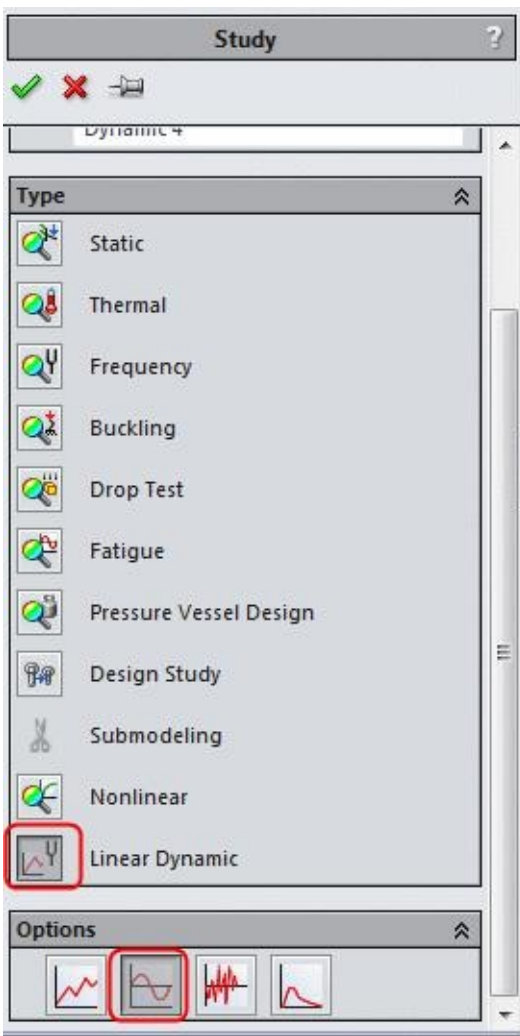


Figure-15. Harmonic Study button

- Click on the **OK** button from the **PropertyManager**. The options related to Harmonic Analysis will be displayed.

Applying Material and Fixture

- Click on the **Apply Material** button from the **Ribbon** and select the **Alloy Steel** material from the **Material** dialog box displayed.
- Click on the **Apply** button and then **Close** button from the **Material** dialog box.
- Click on the down arrow below **Fixtures Advisor** button in the **Ribbon** and select the **Fixed Geometry** button. The **Fixture PropertyManager** will be displayed.
- Select the round bottom face of the bolt as shown in [Figure-16](#).

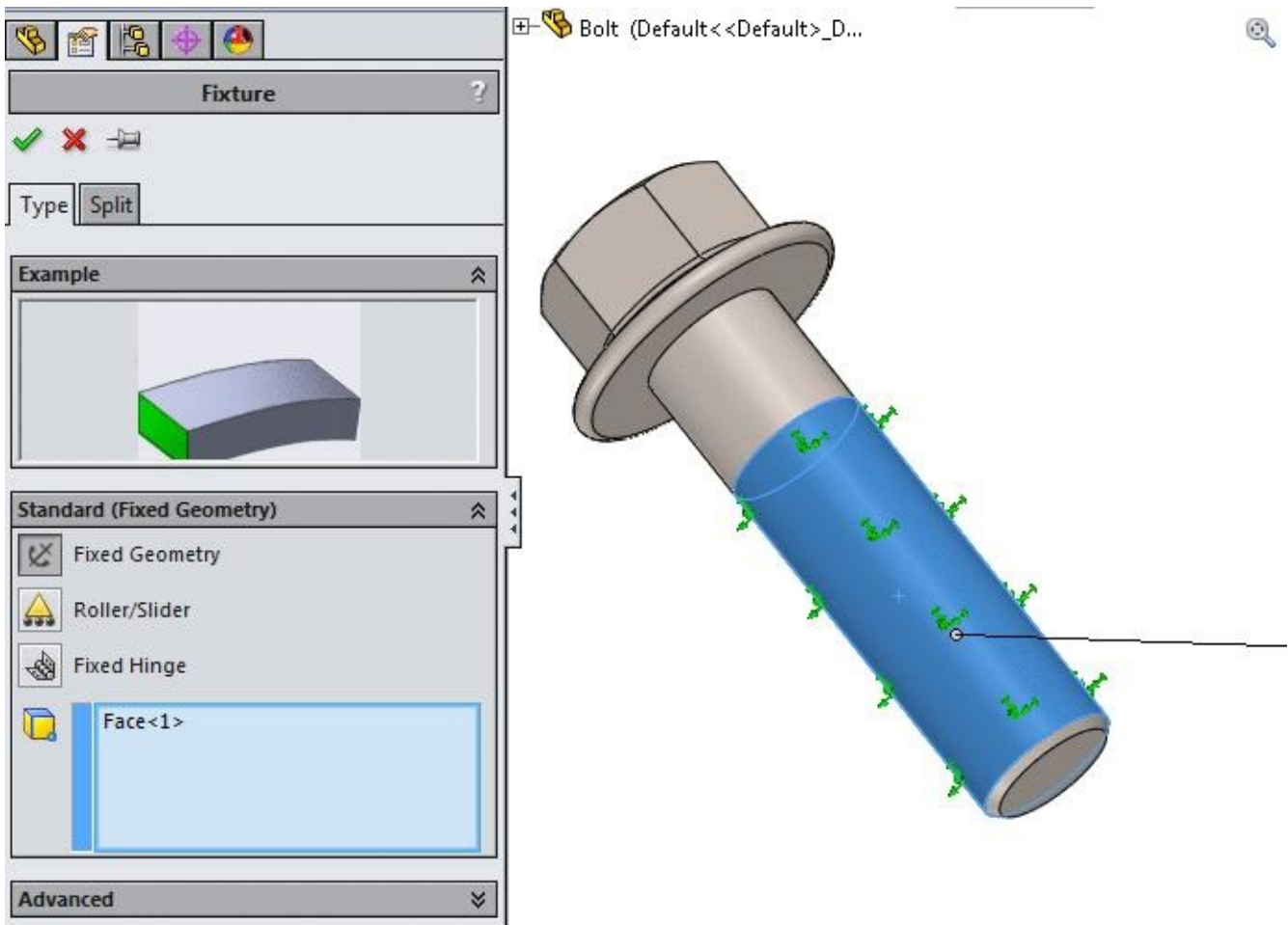


Figure-16. Fixed round face

- Click on the **OK** button from the **PropertyManager** to apply fixture.

Applying Force

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of tools will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed.
- Select the flat face of the bolt as shown in [Figure-17](#).

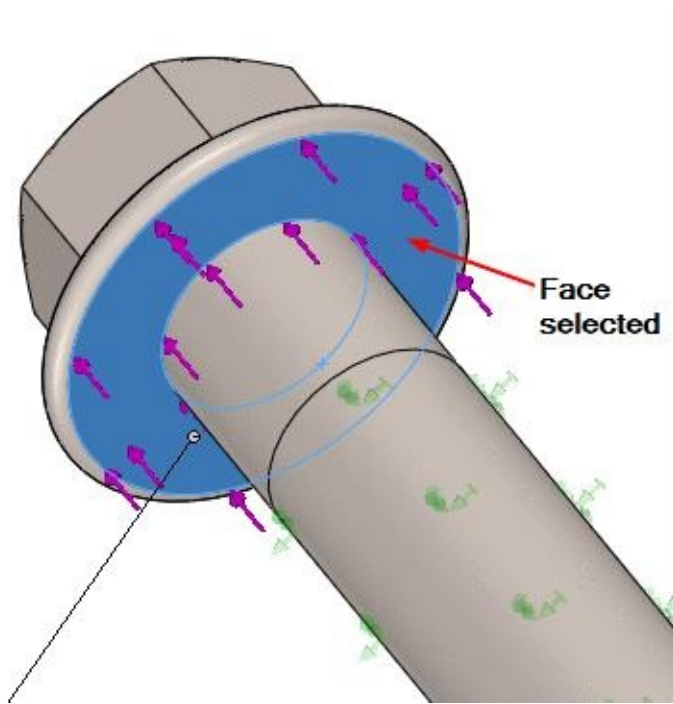


Figure-17. Face selected for applying force

- Click in the **Force Value** edit box in the **PropertyManager** and specify the value as **50 N**.
- Click in the **Phase Angle** edit box in **Phase Angle** rollout if you want to specify phase angle. (We have kept it zero. Remember the formula; $P = A \sin(\omega t + \varphi)$ where φ is phase angle).
- Click on the **OK** button from the **PropertyManager** to apply the force.

Setting Frequency Range

- Right-click on the Harmonic study from the **Analysis Manager** and select the **Properties** option from the shortcut menu; refer to [Figure-18](#). The **Harmonic** dialog box will be displayed; refer to [Figure-19](#).

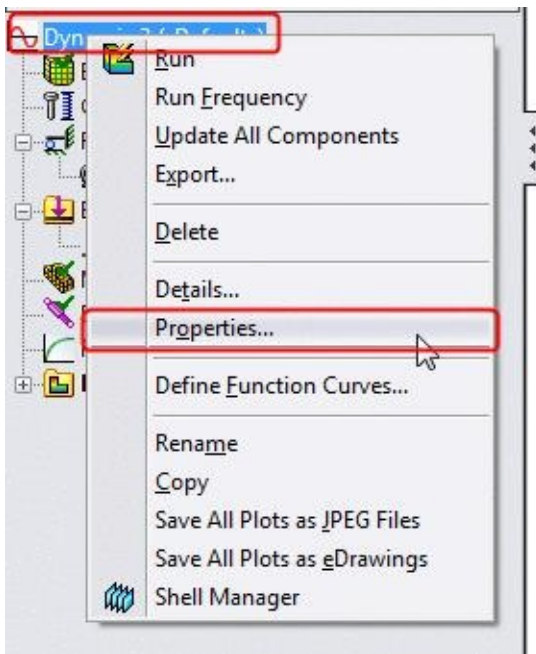


Figure-18. Properties option

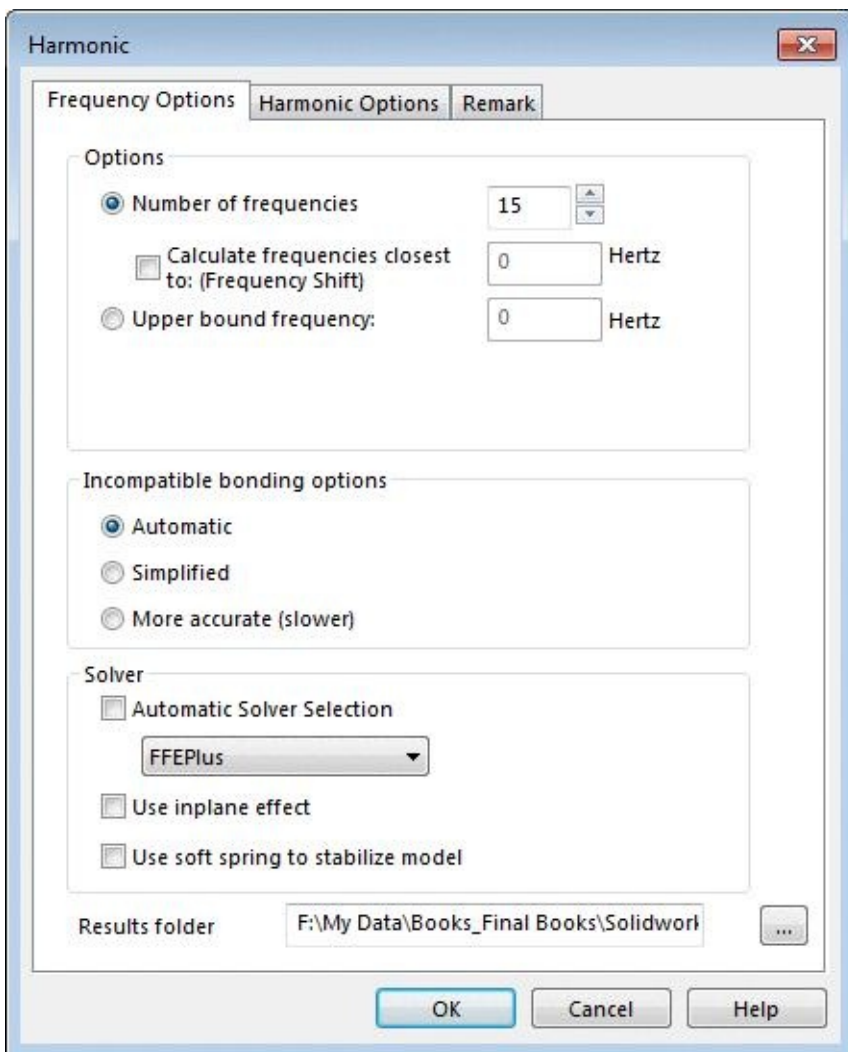


Figure-19. Harmonic dialog box

- Specify the number of frequencies to be tested to **10** by using the **Number of**

frequencies spinner in the Options area of the dialog box.

- Click on the **Harmonic options** tab and specify the **Upper limit** as 5000 Hz; refer to [Figure-20](#).

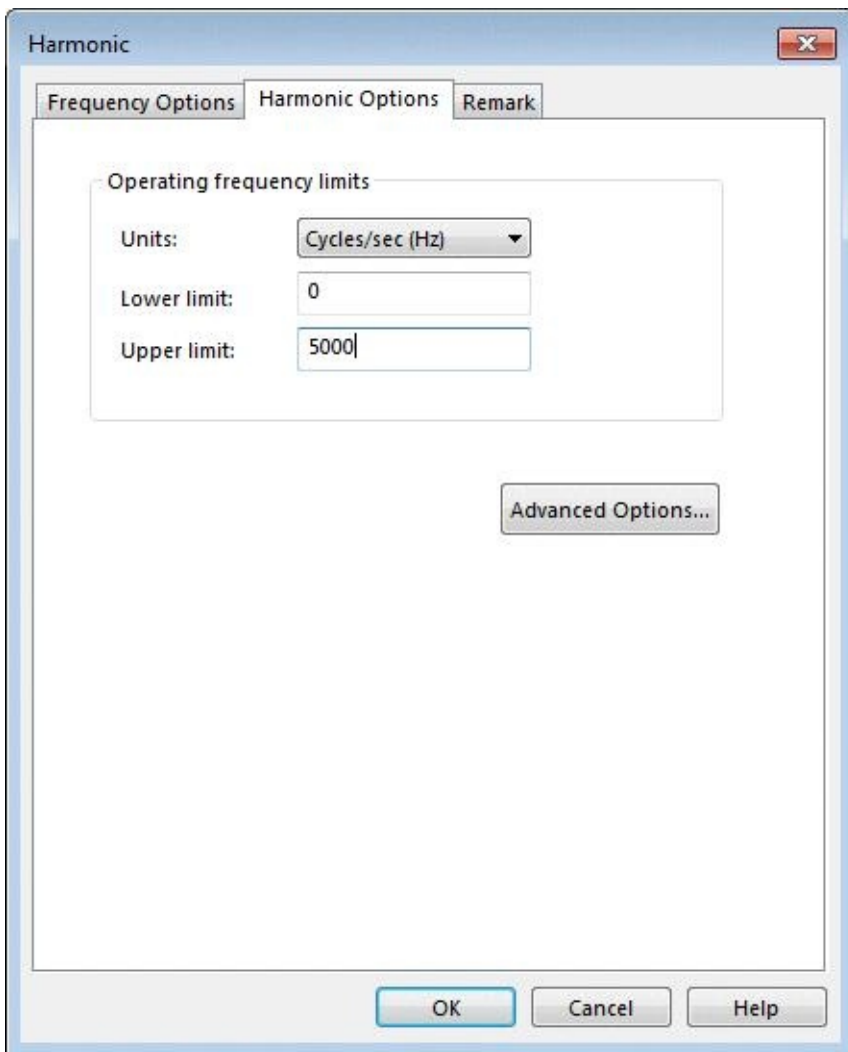


Figure-20. Upper limit for frequencies

- Click on the **OK** button from the dialog box.

Jimmy reaches or not (Running Analysis)

- Click on the **Run This Study** button from the **Ribbon** and wait & watch. The Stress result will be displayed as shown in [Figure-21](#) which concludes that nobody can stop Jimmy. The stress in bolt is well below the yield strength.

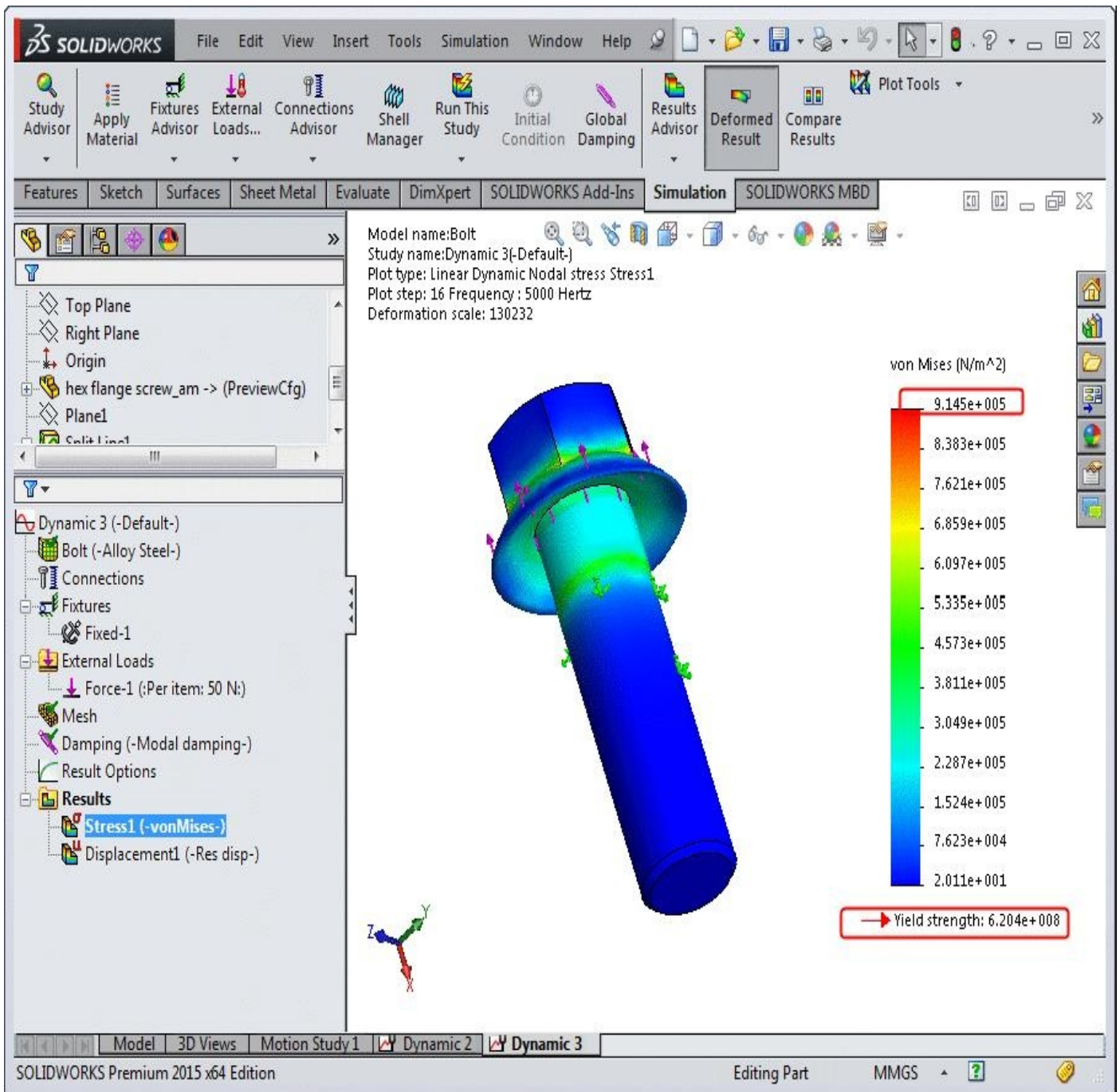


Figure-21. Result of Harmonic study

There are some situations where you need to see the effect of base excitation on the object. In such cases, you might be required to perform the Response Spectrum analysis. The process is given next.

RESPONSE SPECTRUM ANALYSIS

Let me tell you that Jimmy placed his camera in the side box of his motorbike by mistake and the camera has experienced all the jerks provided by road. Let's what happens to the body of his new camera. Assume that the excitation cause on camera body is 2 m/s^2 .

Starting Response Spectrum Analysis

- Open the part on which you want to perform the analysis (obviously camera body!!).
- Click on the down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Linear Dynamic** button and then on the **Response Spectrum Analysis** button from the **Options** rollout; refer to [Figure-22](#).

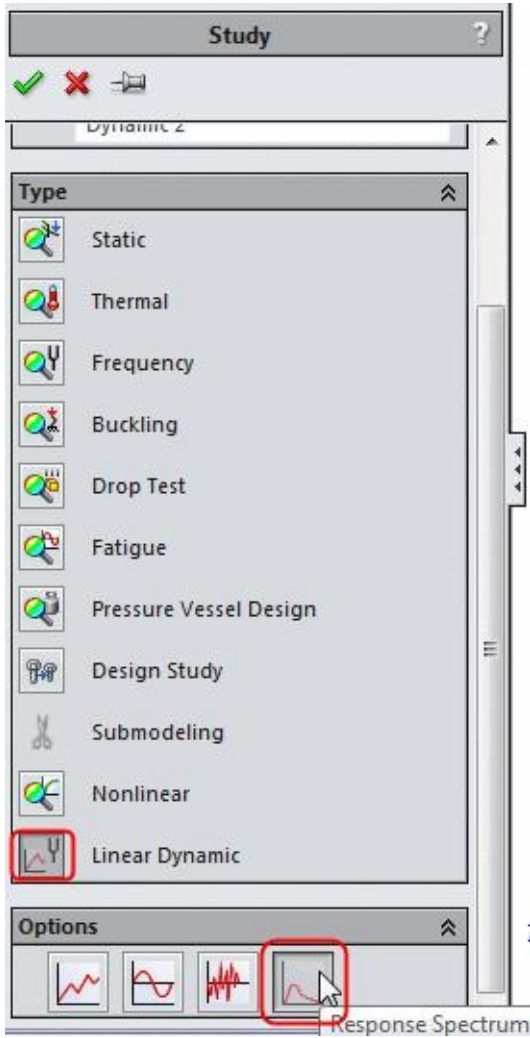


Figure-22. Response Spectrum Analysis button

- Click on the **OK** button from the **PropertyManager**. The options related to Response Spectrum Analysis will be displayed.

Applying Material and Fixture

- Click on the **Apply Material** button from the **Ribbon** and select the **Acrylic (Medium-high impact)** plastic from the **Material** dialog box displayed; refer to [Figure-23](#).

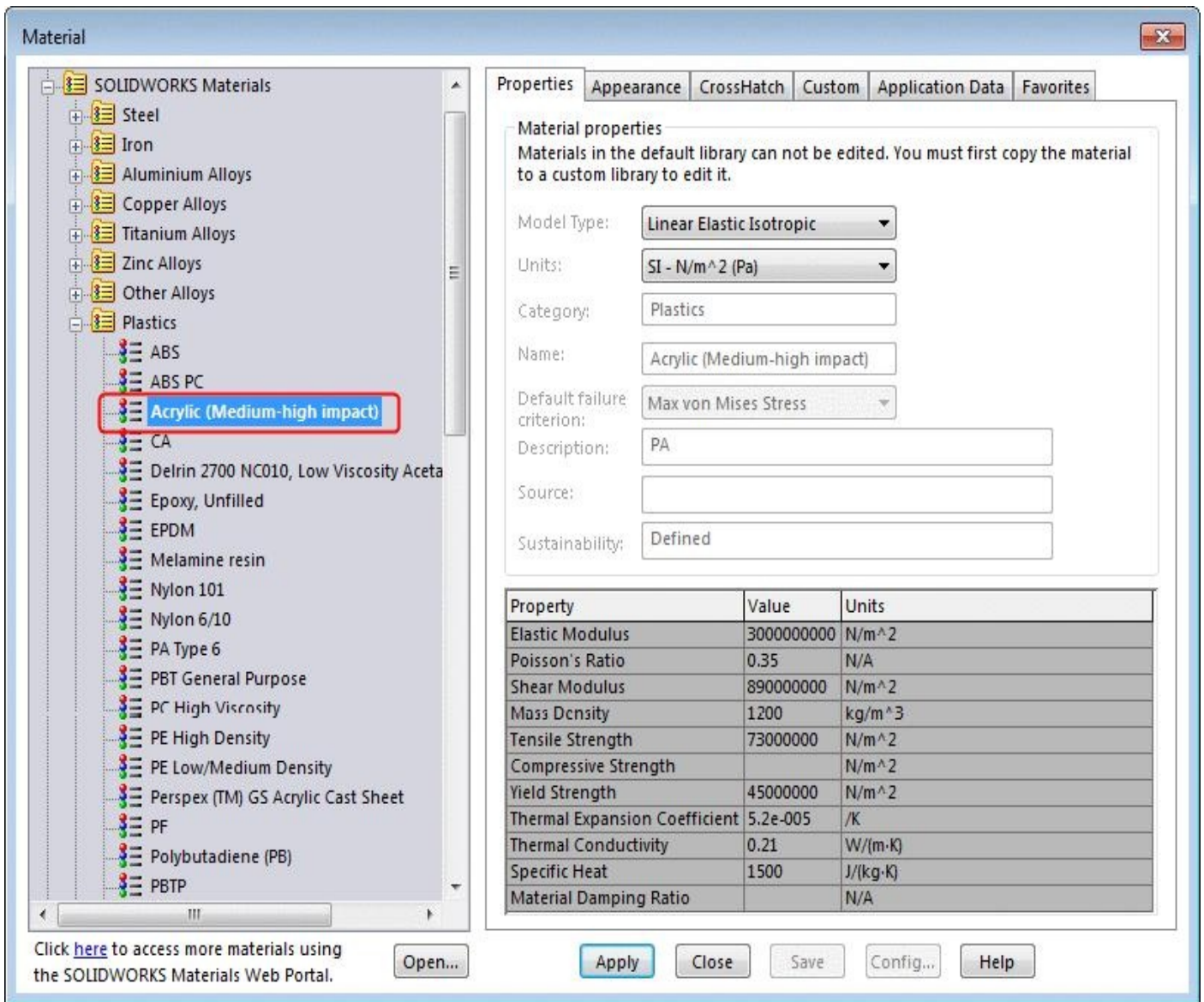


Figure-23. Material dialog box with plastic materials

- Click on the **Apply** button and then **Close** button from the **Material** dialog box.
- Click on the down arrow below **Fixtures Advisor** button in the **Ribbon** and select the **Fixed Geometry** button. The **Fixture PropertyManager** will be displayed.
- Select the bottom face of the camera as shown in [Figure-24](#).

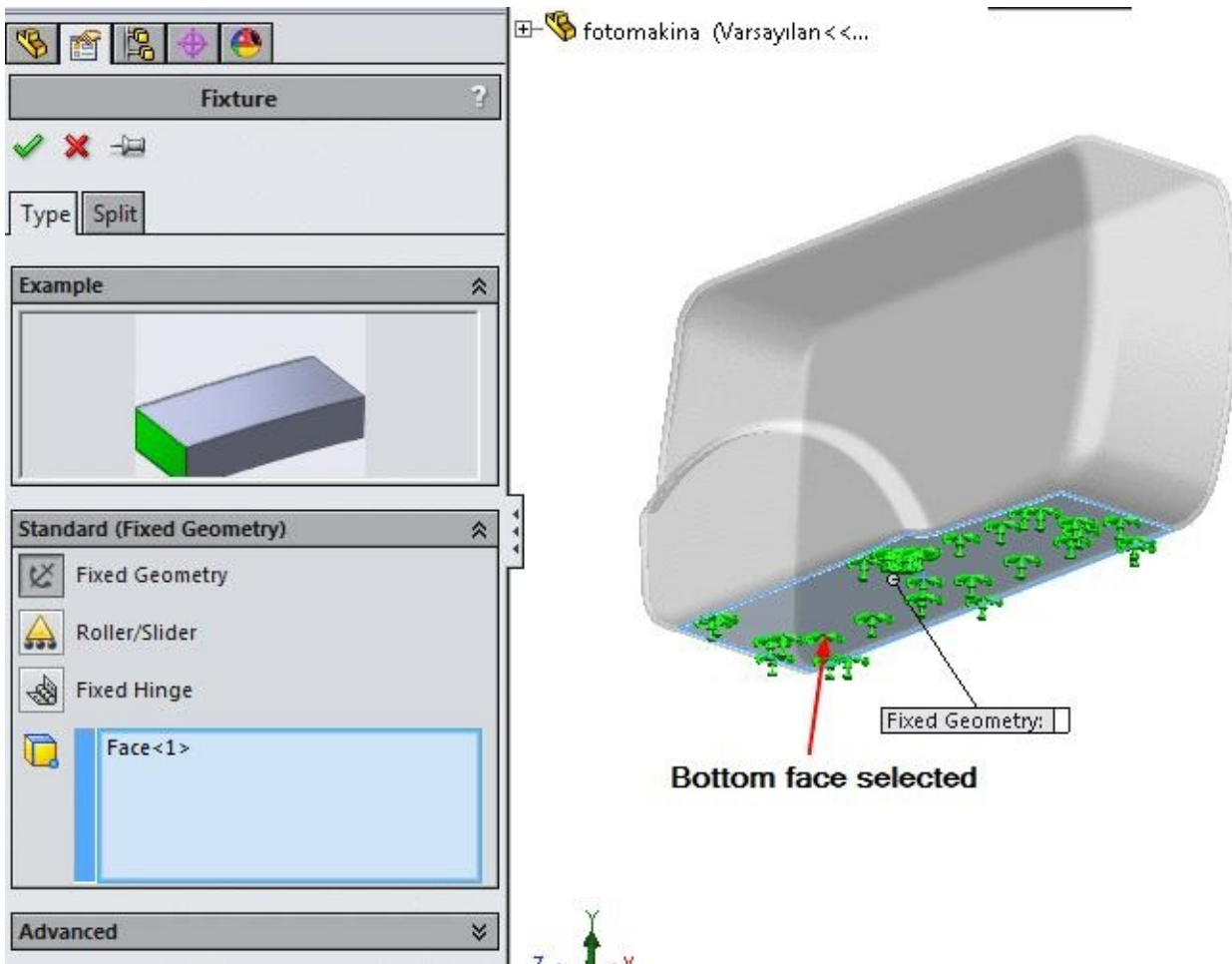


Figure-24. Bottom face of camera body selected

- Click on the **OK** button from the **PropertyManager** to apply fixture.

Applying Excitation

No sir! Excitation is not linked with human emotions here. Excitation is an displacement, velocity, or acceleration caused in the body due to vibrations. There are two tools available to define excitation; refer to [Figure-25](#).

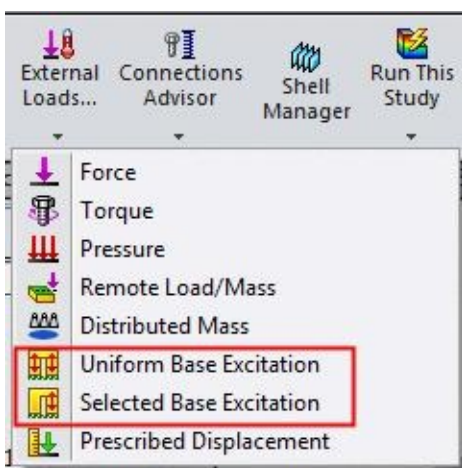


Figure-25. Excitation tools

If you have specified base at more than one faces then you can use the **Selected Base Excitation** tool to specify excitation for individual bases. To specify the same excitation for all the bases, use the **Uniform Base Excitation** tool.

- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. List of tools will be displayed.
- Click on the **Uniform Base Excitation** tool from the list. The **Uniform Base Excitation PropertyManager** will be displayed.
- Select the **Acceleration** radio button from the **Type** rollout and specify **2** in vertical direction; refer to [Figure-26](#).

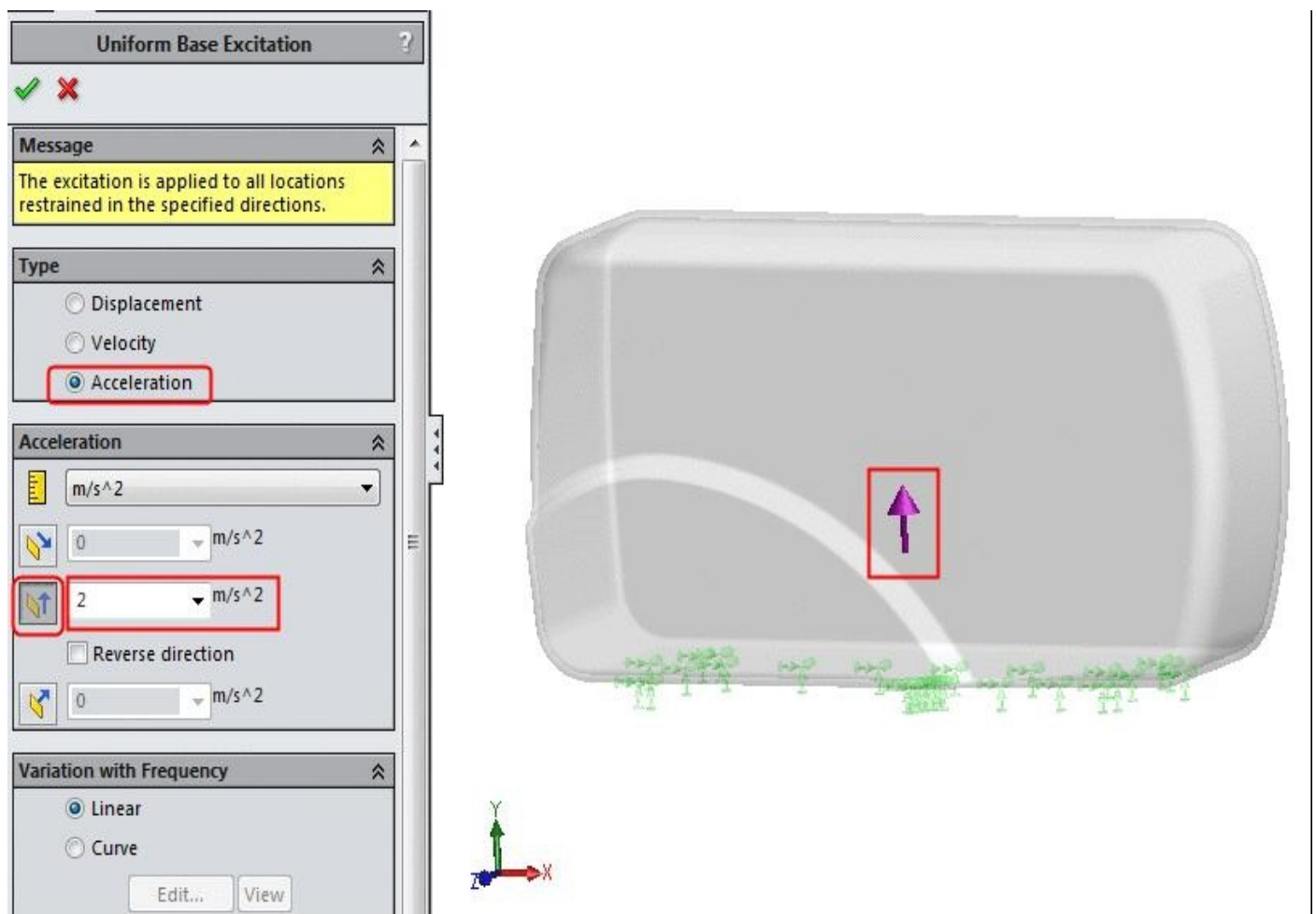


Figure-26. Uniform Base Excitation PropertyManager

- Click on the **OK** button from the **PropertyManager** to apply the excitation.

Now we are ready for the magic. Click on the **Run This Study** button from the **Ribbon** and see whether the camera is in one piece or more pieces; refer to [Figure-27](#).

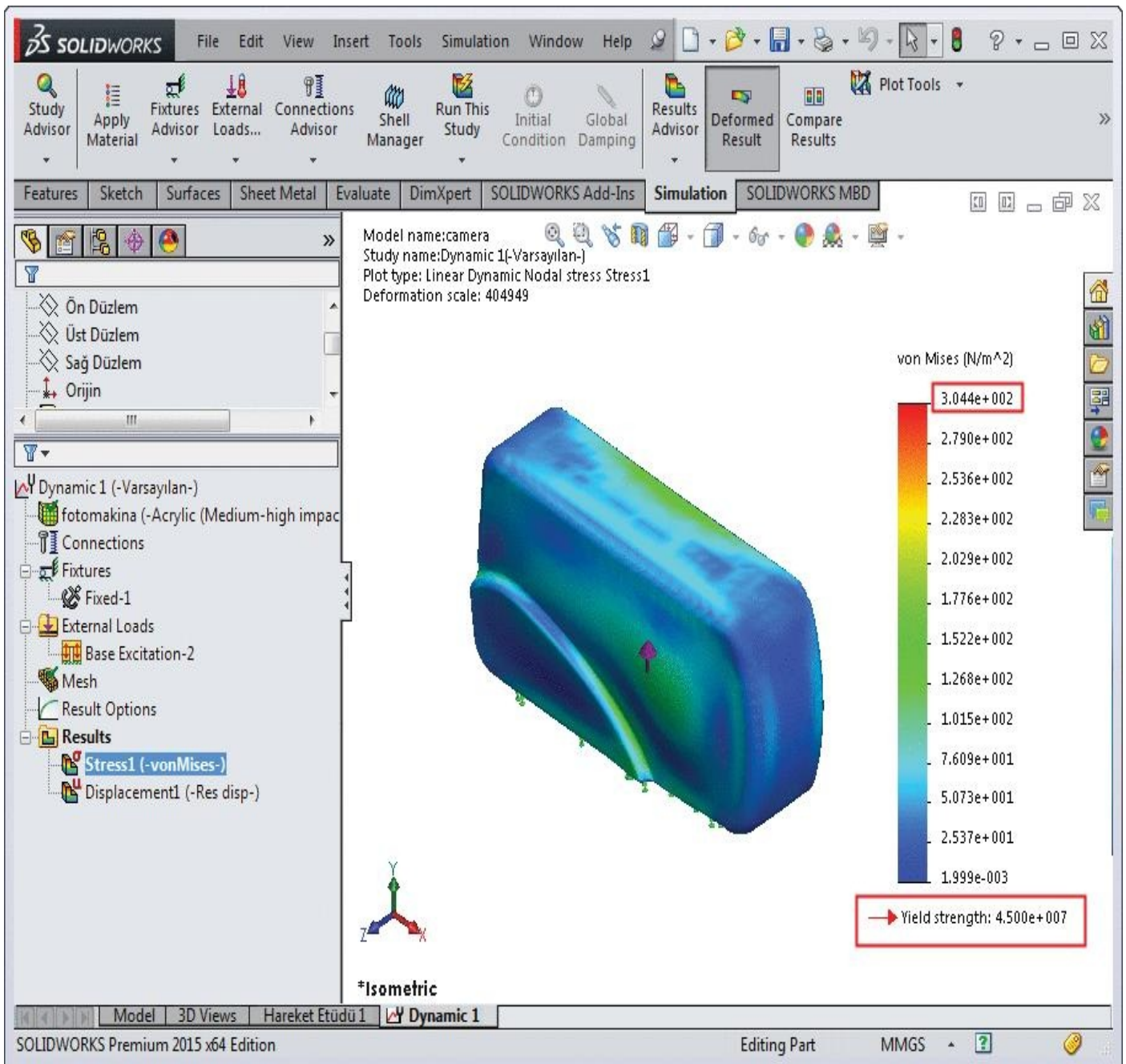


Figure-27. Result of Response Spectrum



Even if you change the value of Base excitation to **20000** m/s², still the camera will be safe. Because the power of acrylic plastic will not let you down. This plastic is used to make bullet proof shields.

PRACTICE 1

In the below problem, you will analyze a driving shaft of automobile for its ability to sustain the torque provided by engine. Note that the driver of this vehicle is rash driver and he really don't care about his machine. So, you might need to perform the random

vibration analysis. The parameters for the analysis are given in [Figure-28](#). The model of this problem is available in resource kit.

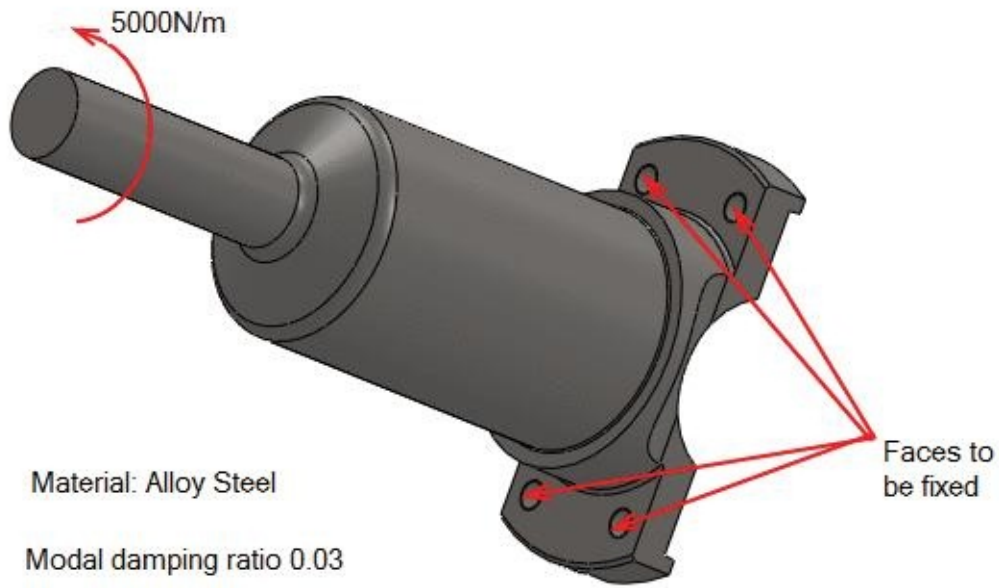


Figure-28. Parameters for analysis

THERMAL ANALYSIS

CHAPTER 8



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Types of Thermal Analysis*
- *Starting Thermal Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

INTRODUCTION

Thermal analysis is a method to check the distribution of heat over a body due to applied thermal loads. Note that thermal energy is dynamic in nature and is always flowing through various mediums. There are three mechanisms by which the thermal energy flows:

- Conduction
- Convection
- Radiation

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.

The output from a thermal analysis can be given by:

1. Temperature distribution.
2. Amount of heat loss or gain.
3. Thermal gradients.
4. Thermal fluxes.

This analysis is used in many engineering industries such as automobile, piping, electronic, power generation, and so on.

Important terms related to Thermal Analysis

Before conducting thermal analysis, you should be familiar with the basic concepts and terminologies of thermal analysis. Following are some of the important terms used in thermal analysis:

Heat Transfer Modes

Whenever there is a difference in temperature between two bodies, the heat is transferred from one body to another. Basically, heat is transferred in three ways: Conduction, Convection, and Radiation.

Conduction

In conduction, the heat is transferred by interactions of atoms or molecules of the material.

For example, if you heat up a metal rod at one end, the heat will be transferred to the other end by the atoms or molecules of the metal rod.

Convection

In convection, the heat is transferred by the flowing fluid. The fluid can be gas or liquid. Heating up water using an electric water heater is a good example of heat convection. In this case, water takes heat from the heater.

Radiation

In radiation, the heat is transferred in space without any matter. Radiation is the only heat transfer method that takes place in space. Heat coming from the Sun is a good example of radiation. The heat from the Sun is transferred to the earth through radiation.

Thermal Gradient

The thermal gradient is the rate of increase in temperature per unit depth in a material.

Thermal Flux

The Thermal flux is defined as the rate of heat transfer per unit cross-sectional area. It is denoted by q .

Bulk Temperature

It is the temperature of a fluid flowing outside the material. It is denoted by T_b . The Bulk temperature is used in convective heat transfer.

Film Coefficient

It is a measure of the heat transfer through an air film.

Emissivity

The emissivity of a material is the ratio of energy radiated by the material to the energy radiated by a black body at the same temperature. Emissivity is the measure of a material's ability to absorb and radiate heat. It is denoted by e . Emissivity is a numerical value without any unit. For a perfect black body, $e = 1$. For any other material, $e < 1$.

Stefan–Boltzmann Constant

The energy radiated by a black body per unit area per unit time divided by the fourth power of the body's temperature is known as the Stefan-Boltzmann constant. It is denoted by σ .

Thermal Conductivity

The thermal conductivity is the property of a material that indicates its ability to conduct heat. It is denoted by K .

Specific Heat

The specific heat is the amount of heat required per unit mass to raise the temperature of the body by one degree Celsius. It is denoted by c .

THERMAL LOADS

To start the thermal analysis, click on the down arrow below **Study Advisor**. Select the **New Study** button from the tools displayed. The **Study PropertyManager** will be displayed. Click on the **Thermal** button from the **Type** rollout and click on the **OK** button from the **PropertyManager**. The tools related to thermal analysis will be displayed. Before performing thermal analysis, we will get familiar with various options required specifically for thermal analysis. One of the major player in thermal analysis is the thermal load. To apply the thermal loads, options are available in the **Thermal Loads** drop-down; refer to [Figure-1](#). The options in this drop-down are discussed next.

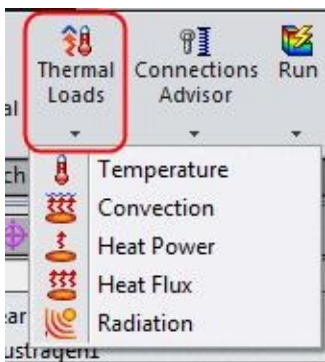


Figure-1. Thermal Loads drop-down

Temperature

This tool is used to specify temperature of selected face/faces. To apply temperature follow the steps given next.

- Click on the **Temperature** tool from the list of tools displayed on click down arrow below **Thermal Loads** in the **Ribbon**. The **Temperature PropertyManager** will be displayed as shown in [Figure-2](#).

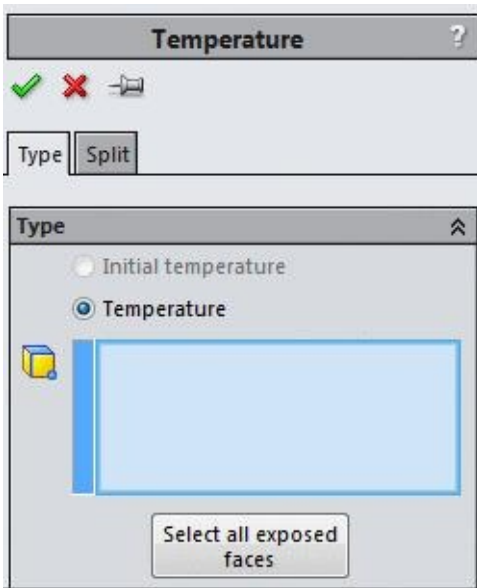


Figure-2. Temperature PropertyManager

- Select the face on which you want to set the temperature.
- Click in the **Temperature** edit box and specify the desired value.
- If you want to change the unit of temperature from the default Kelvin unit, then click on the **Kelvin (K)** option next to **Temperature** edit box. The list of options related to temperature units will be displayed.
- Select the desired unit from the list.
- If you are performing the transient thermal study, then the **Initial temperature** radio button will be active; refer to [Figure-3](#).

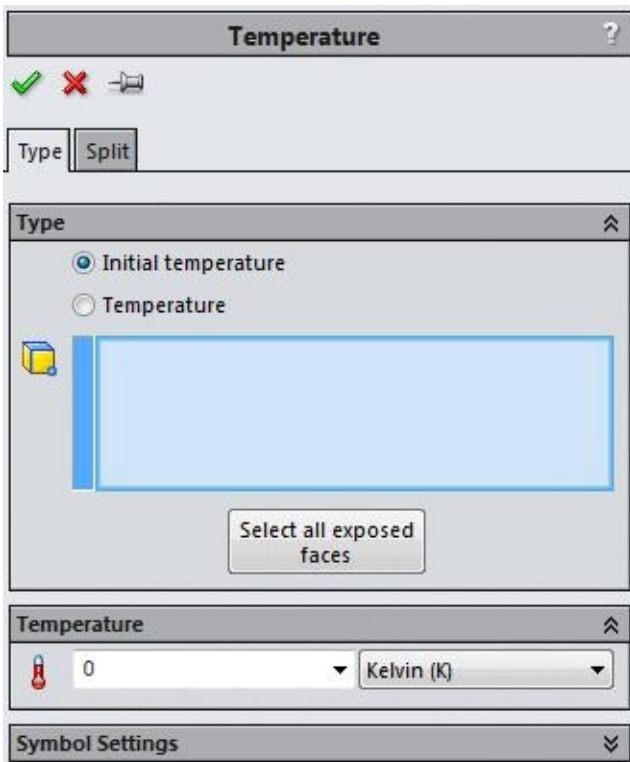


Figure-3. Temperature PropertyManager for transient study

How to start Transient Thermal Analysis? After starting the thermal analysis, right-click on the analysis name in the **Analysis Manager**; refer to [Figure-4](#). The options related to analysis will be displayed in the shortcut menu. Select the **Properties** option from the menu. The **Thermal** dialog box will be displayed; refer to [Figure-5](#). Select the **Transient** radio button from the **Solution Type** area of the dialog box. The options related to analysis time will be displayed. Set the desired time duration and increment for the analysis by using the edit boxes. The other options in this dialog box have already been discussed in previous chapters. Click on the **OK** button from the dialog box to apply the settings.

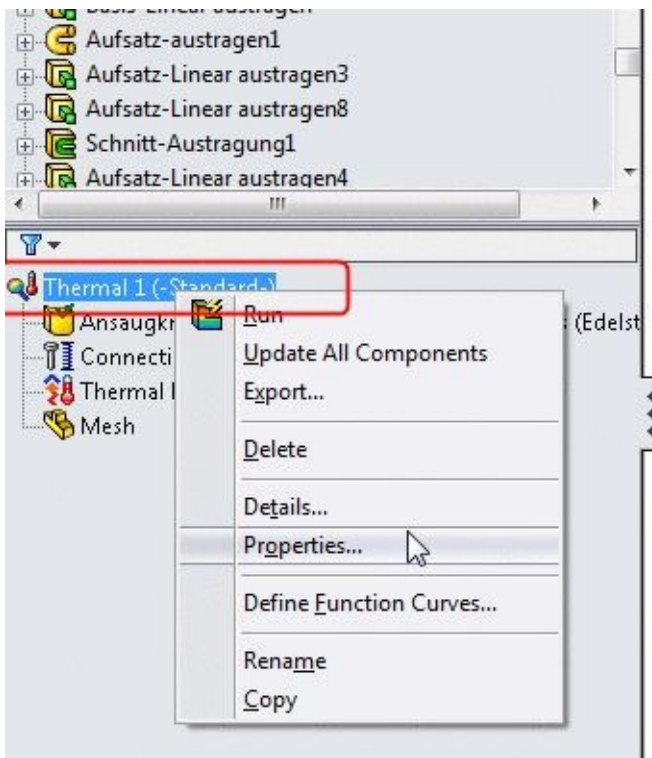


Figure-4. Analysis Properties shortcut menu

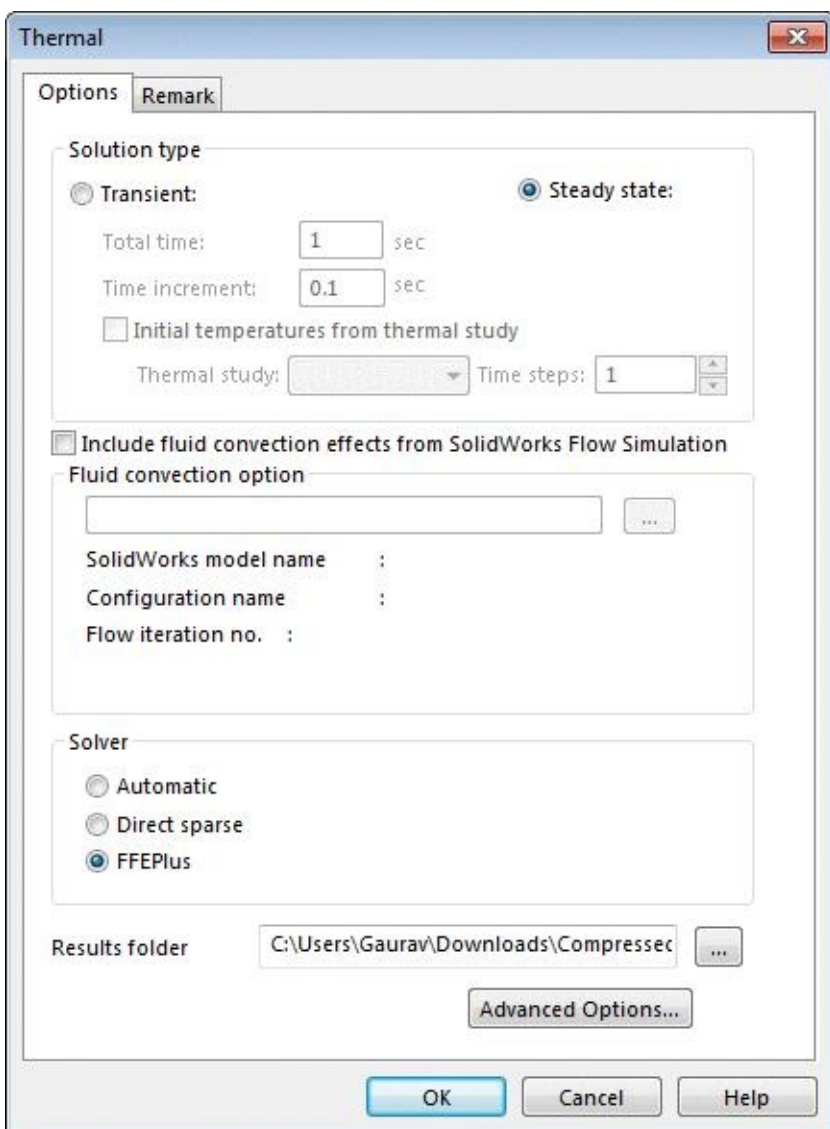


Figure-5. Thermal dialog box

- Using this option, you can specify the starting temperature of the face. Although it will change automatically under the application of heat flux. You will learn more about this tool later while performing the analysis.
- If you select the **Temperature** radio button for a transient thermal study then the **Variation with Time** rollout is also displayed in the **PropertyManager**; refer to [Figure-6](#).

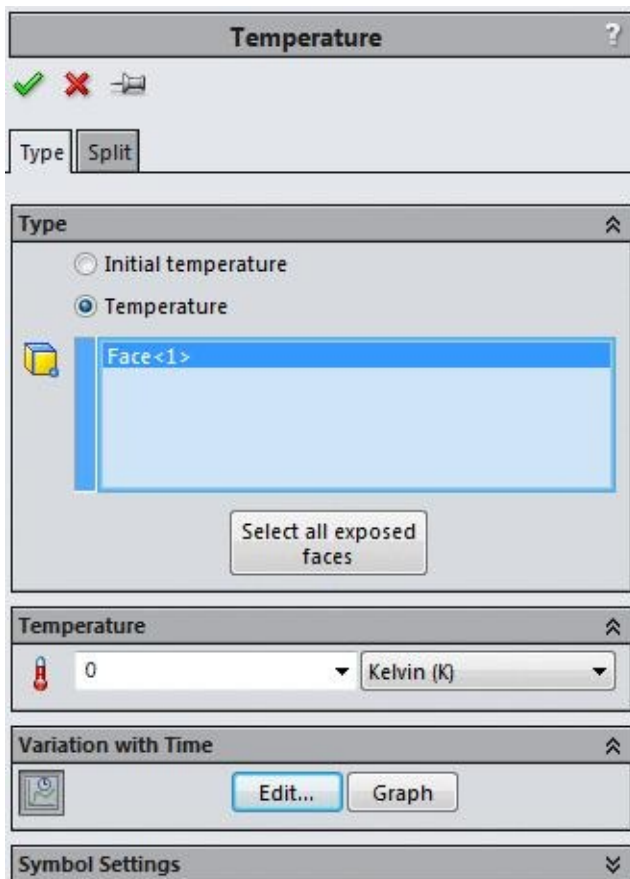


Figure-6. Temperature PropertyManager with time varying temperature

- The options in the **Variation with Time** rollout are same as discussed in previous chapters.
- If you want to select the all outer faces of the model at once, then click on the **Select all exposed faces** button from the **PropertyManager**.
- After specifying the desired parameters, click on the **OK** button from the dialog box.

Convection

Convection is the phenomena according to which heat is transferred through a medium like air, fluid or solid. Every medium has its limitation to transfer the heat energy. This limitation is mathematically represented by **Convection Heat coefficient $Q_{\text{convection}}$** .

$$Q_{\text{convection}} = hA(T_s - T_f)$$

Here, **h** is heat transfer coefficient has unit W/m².k

T_s is the temperature of the surface.

T_f is the temperature of the surrounding fluid.

A is the area of surface.

In SolidWorks Simulation, we need to specify the **Q_{convection}** for the convection phenomena. The steps to apply the heat convection are given next.

- Click on the down arrow below the **Thermal Loads** from the **Ribbon**. The list of tools will be displayed.
- Select the **Convection** button from the list of tools. The **Convection PropertyManager** will be displayed as shown in [Figure-7](#).

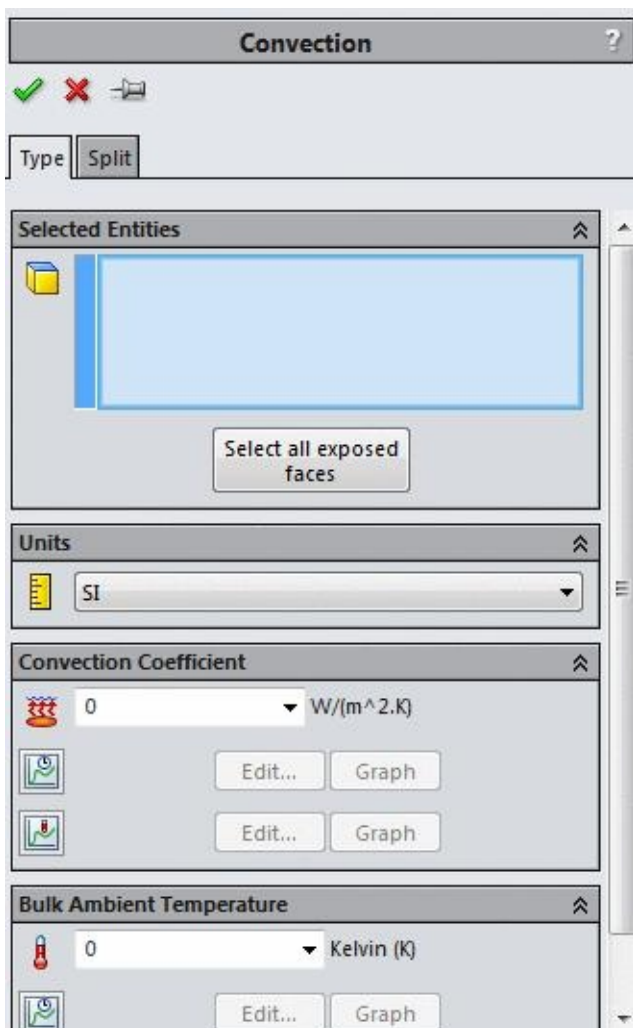


Figure-7. Convection PropertyManager

- Select the face from which the convection is occurring.
- Click in the **Convection Coefficient** edit box in the **Convection Coefficient** rollout and specify the desired value in it.
- Click in the **Bulk Ambient Temperature** edit box in the **Bulk Ambient Temperature** rollout and specify the surrounding temperature for the analysis.
- After specifying the desired parameters, click on the **OK** button from the **PropertyManager** to apply the heat convection.

Heat Power

This is the energy applied on the selected face in the form of heat. You can specify the heat power applied on the face by using the Heat Power button. The steps to use this button are given next.

- Click on the down arrow below the **Thermal Loads** button in the **Ribbon**. The list of tools will be displayed.
- Click on the **Heat Power** button from the list. The **Heat Power PropertyManager** will be displayed; refer to [Figure-8](#).

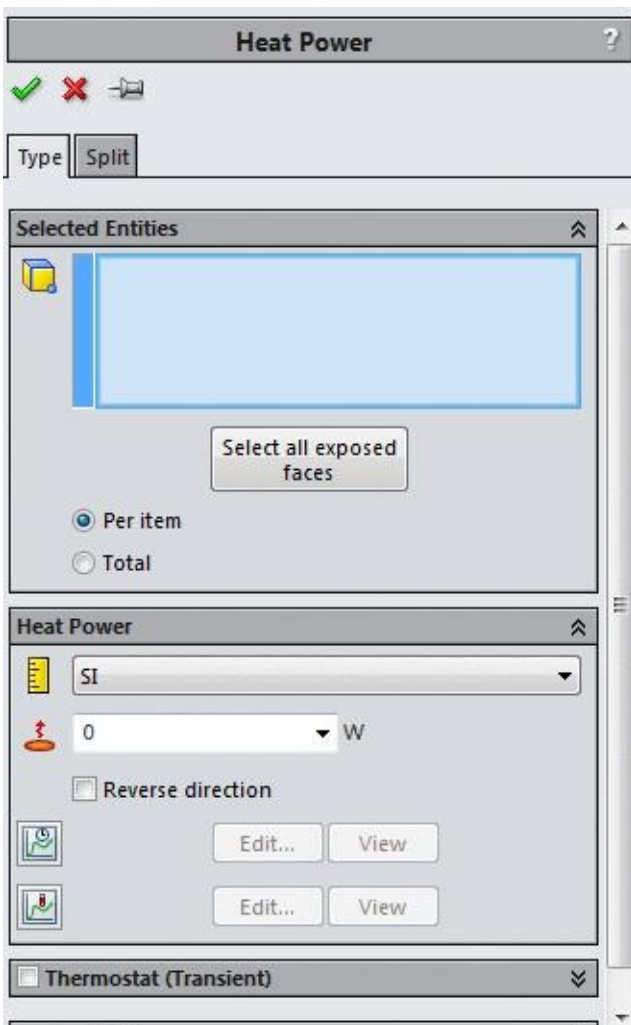


Figure-8. Heat Power PropertyManager

- Select the face on which you want to apply the heat.
- Click in the **Heat Power** edit box in the **Heat Power** rollout in **PropertyManager** and specify the desired value of heat power.
- If you are performing transient thermal study then you can apply a thermostat condition by select the check box on the **Thermostat (Transient)** rollout.
- On selecting the check box of **Thermostat (Transient)** rollout, the expanded rollout is displayed; refer to [Figure-9](#).

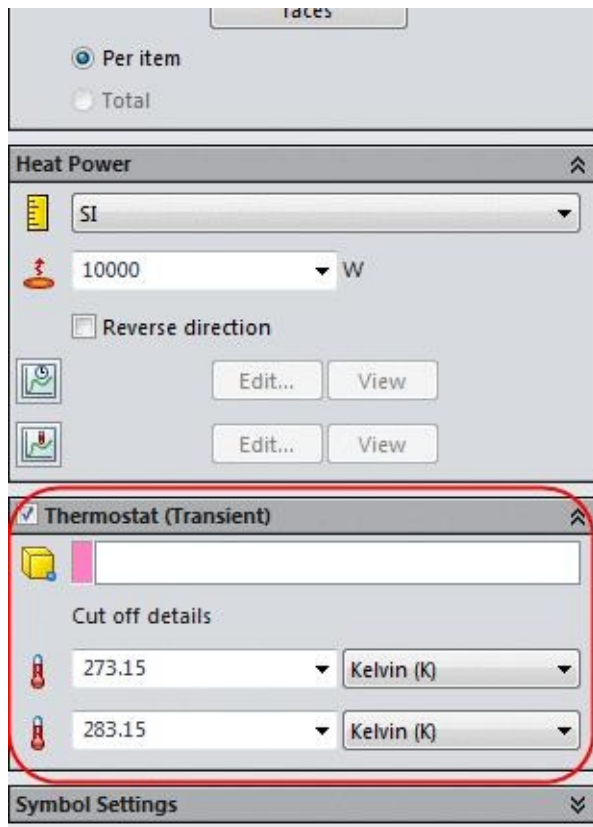


Figure-9. Thermostat rollout

- Click in the pink selection box in the rollout and select the vertex that you want to select as a sensor for the thermostat.
- In the first edit box below the pink selection box, you can specify the minimum temperature for starting up the heat power.
- In the next edit box, you can specify the maximum temperature up to which the power source will provide the heat.
- After specifying the desired parameters, click on the **OK** button from the **PropertyManager**.

Heat Flux

Heat flux is the heat energy applied per unit area of the surface. The steps to apply heat flux on the model are given next.

- Click on the down arrow below **Thermal Loads** button in the **Ribbon**. The list of tools will be displayed.
- Click on the **Heat Flux** button from the list. The **Heat Flux PropertyManager** will be displayed as shown in [Figure-10](#).

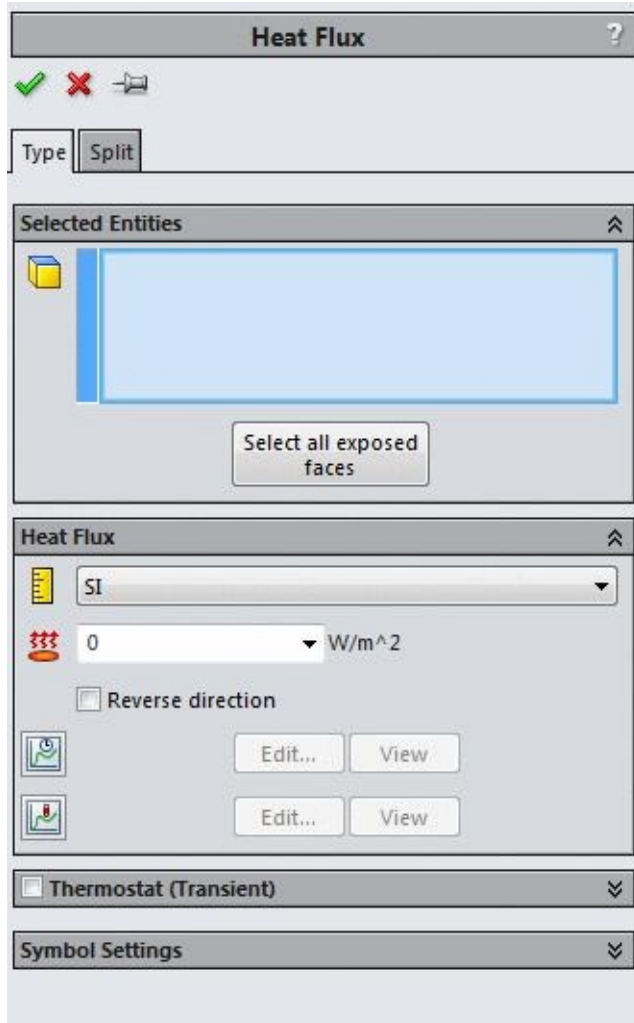


Figure-10. Heat Flux PropertyManager

- The options in this **PropertyManager** are same as in **Heat Power PropertyManager** and can be used in the same way.

Radiation

Thermal radiation is the thermal energy emitted by bodies in the form of electromagnetic waves because of their temperature. All bodies with temperatures above the absolute zero emit thermal energy. Because electromagnetic waves travel in vacuum, no medium is necessary for radiation to take place. Thermal radiation can occur between **ambient temperature surrounding and surface of body** or between **surface of one body and**

surface of other body. The steps to specify radiation are given next.

- Click on down arrow below **Thermal Loads** button. The list of tools will be displayed.
- Click on the **Radiation** button from the list. The **Radiation PropertyManager** will be displayed as shown in [Figure-11](#).

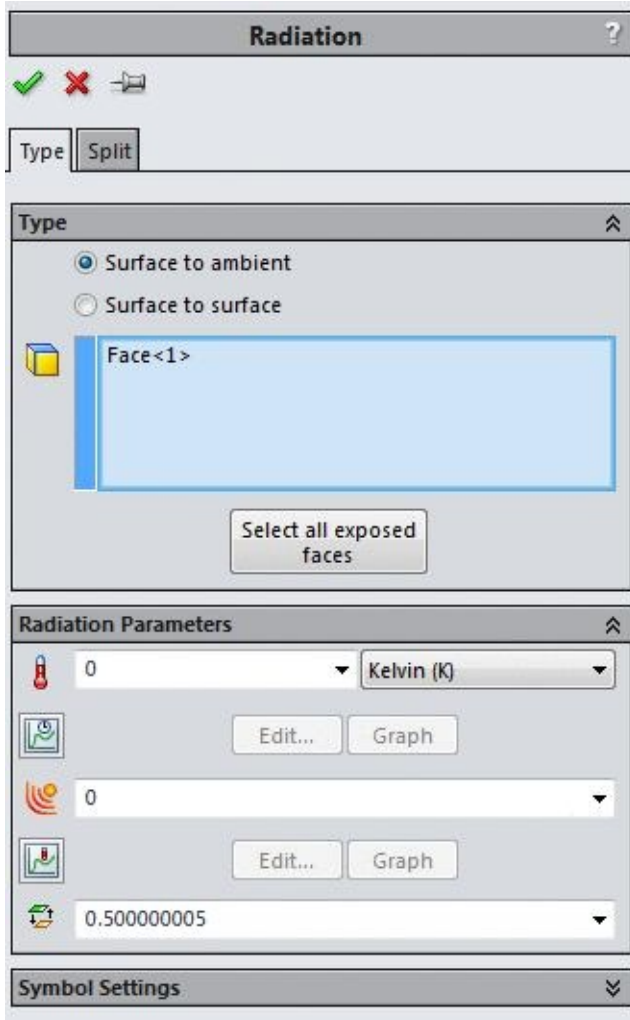


Figure-11. Radiation PropertyManager

- Select the face from which the radiation is occurring.
- Click in the first edit box in the **Radiation Parameters** rollout and specify the ambient temperature if you have selected the **Surface to ambient** radio button from the **Type** rollout.
- If you have selected the **Surface to surface** radio button from the **Type** rollout and selected the **Open system** check box from the **Radiation Parameters** rollout then also you can specify the ambient temperature in the first edit box in the **Radiation Parameters** rollout.
- Click in the **Emissivity** edit box and specify the emission coefficient for radiation.
- After specifying the desired parameters, click on the **OK** button from the **PropertyManager**.

CONTACT SET

We have used connections between various parts of assembly while performing structural analyses. In the same way, we are also required to specify connections between various parts of assembly in terms of thermal properties. For thermal analyses, we can specify the following connections:

- Thermal Resistance
- Bonded
- Insulated

The tools to specify these connections are available in the **Connections Advisor** drop-down. Click on the Contact Set tool from the drop-down. The **Contact Sets PropertyManager** will be displayed as shown in [Figure-12](#).

Select the desired contact type from the drop-down in the **Type** rollout and select the faces to apply the thermal contact type. In the same way, you can use the **Component Contact** button to apply the connection between two parts.

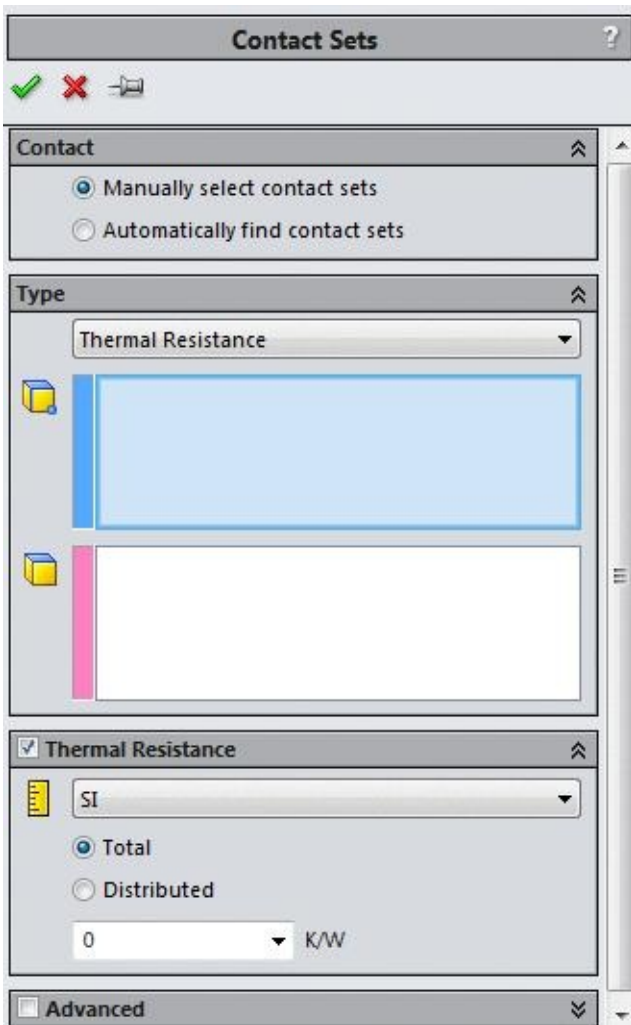


Figure-12. Contact Sets PropertyManager

PRACTICAL ON STEADY STATE THERMAL ANALYSIS

In this tutorial, we will perform a steady state thermal analysis on a real world model and check the distribution of temperature/heat over the surface of the model. Note that model for this practical is available in the resource kit.

Starting the analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Thermal** button then click on the **OK** button from the **PropertyManager**. The tools related to thermal analysis will be displayed.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed.
- Expand the **SolidWorks Materials** node and then the **Aluminium Alloys** node in the left of the dialog box.
- Browse through the materials and select the **C355.0-T61 Permanent Mold cast (SS)** material; refer to [Figure-13](#).
- Click on the **Apply** button and then click on the **Close** button from the dialog box to exit the dialog box.

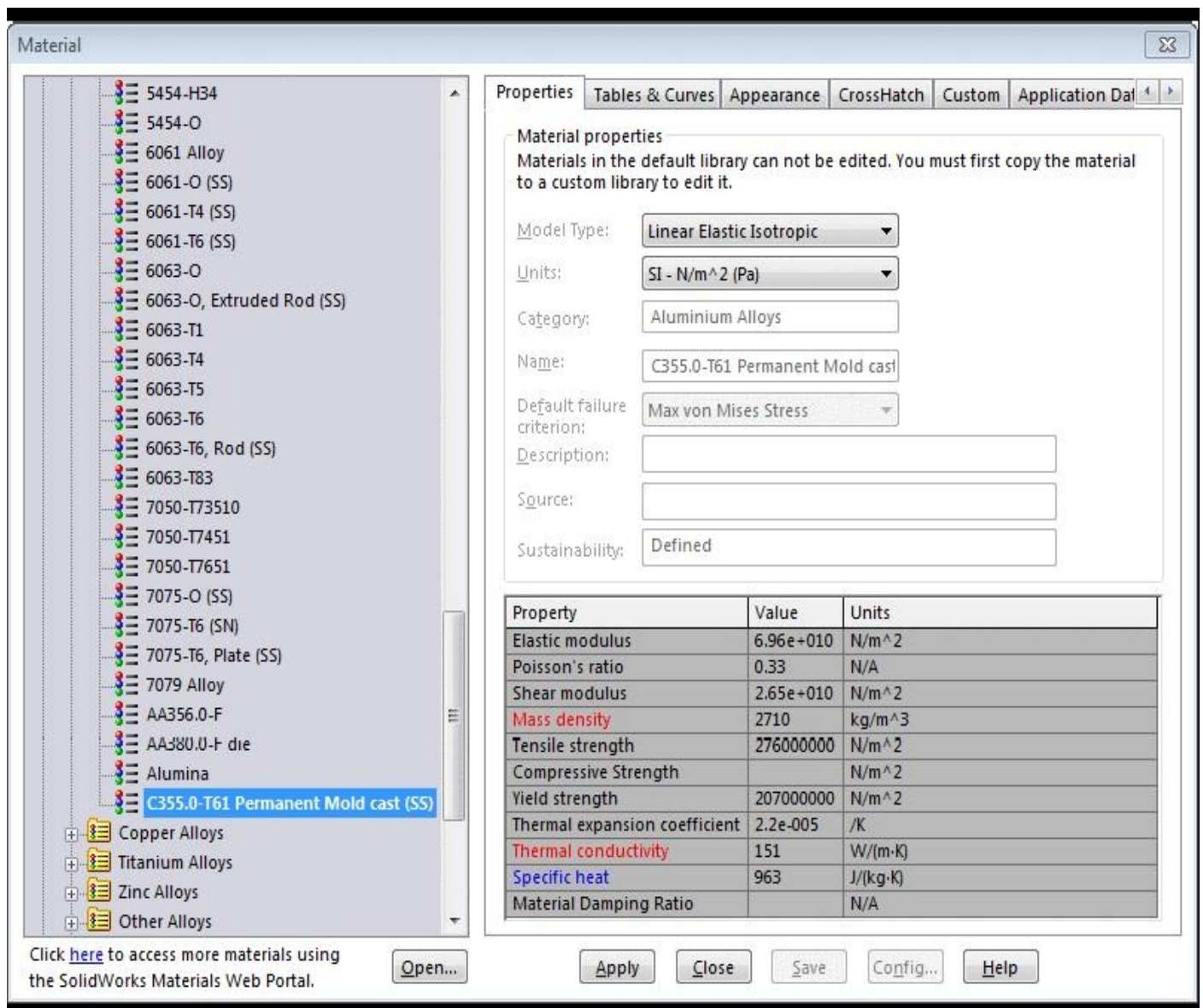


Figure-13. Material dialog box

Applying temperature on the face

- Click on the down arrow below **Thermal Loads** button and select the **Temperature** button from the list. The **Temperature PropertyManager** will be displayed as shown in [Figure-14](#).

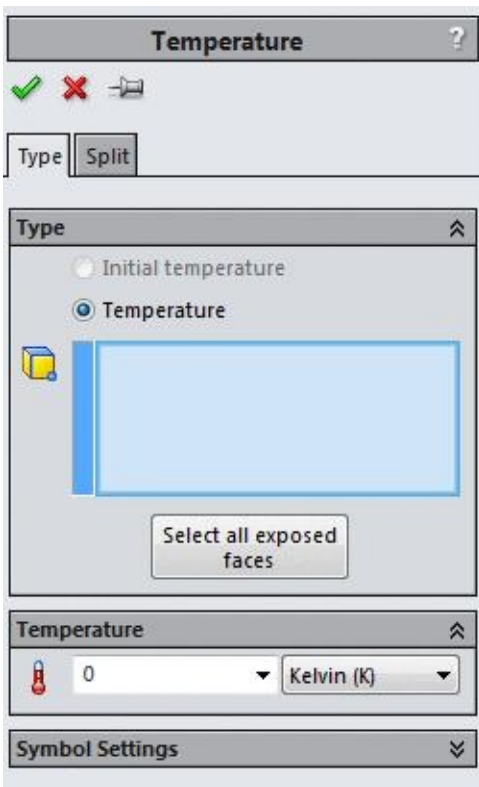


Figure-14. Temperature PropertyManager

- Select the face of the model as shown in [Figure-15](#).

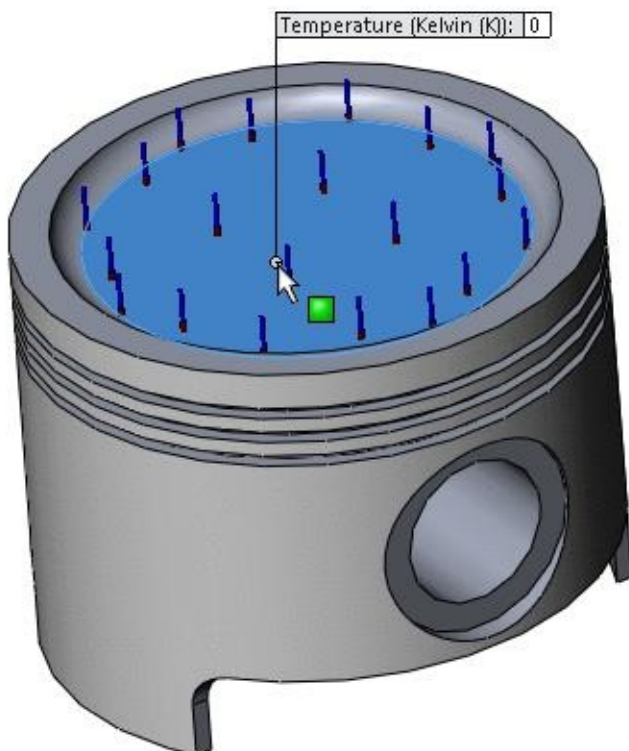


Figure-15. Face of model to be selected

- Click on the drop-down next to **Temperature** edit box in the **Temperature** rollout. List of units for temperature will be displayed.
- Select the **Celsius (°C)** option from the list.
- Click in the **Temperature** edit box and specify the value as **700**.
- Click on the **OK** button from the **PropertyManager** to apply the temperature.

Applying heat generated by gas combustion on the head of the piston

- Click on the down arrow of **Thermal Loads** in the **Ribbon** and select the **Heat Power** button from the list displayed. The **Heat Power PropertyManager** will be displayed; refer to [Figure-16](#).

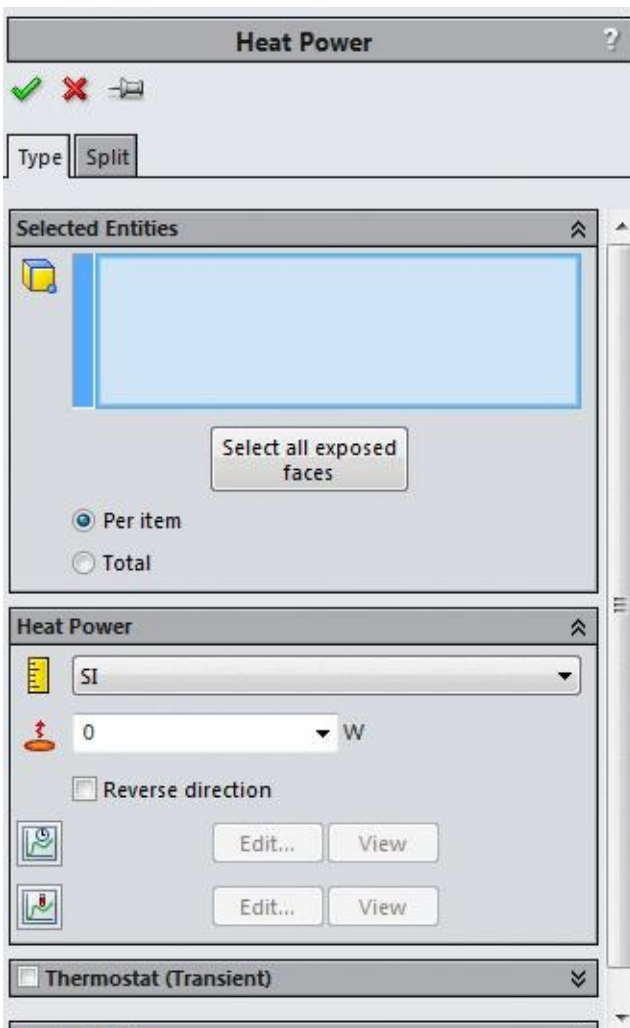


Figure-16. Heat Power PropertyManager

- Click on the **Total** radio button in the **Selected Entities** rollout.
- Select the head faces of the piston as shown in [Figure-17](#).

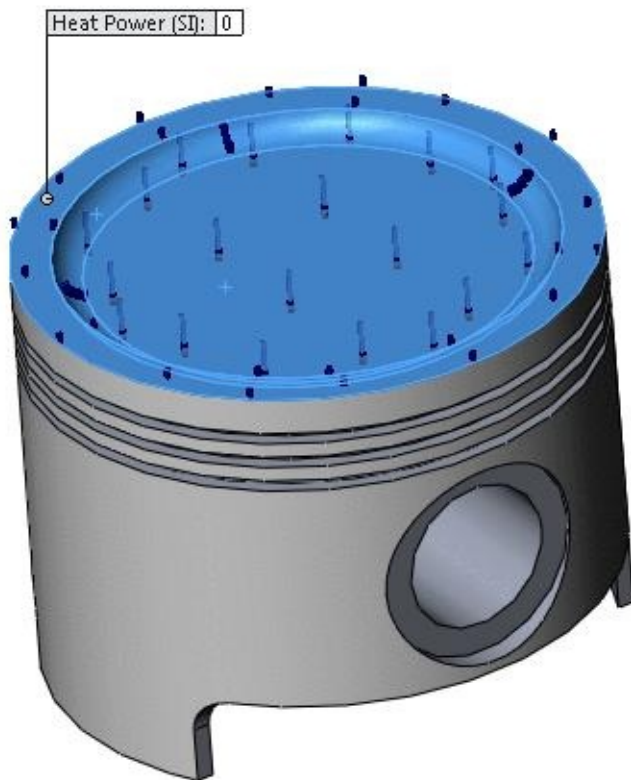


Figure-17. Faces to be selected

- Click in the **Heat Power** edit box in the **Heat Power** rollout. Specify the total heat power as **20000**.
- Click on the **OK** button from the **PropertyManager** to apply the heat.

Running the Analysis

- Click on the Run button from the Ribbon. The analyzing of problem will start.
- After the analysis is complete. The solution of the analysis will be displayed; refer to [Figure-18](#).

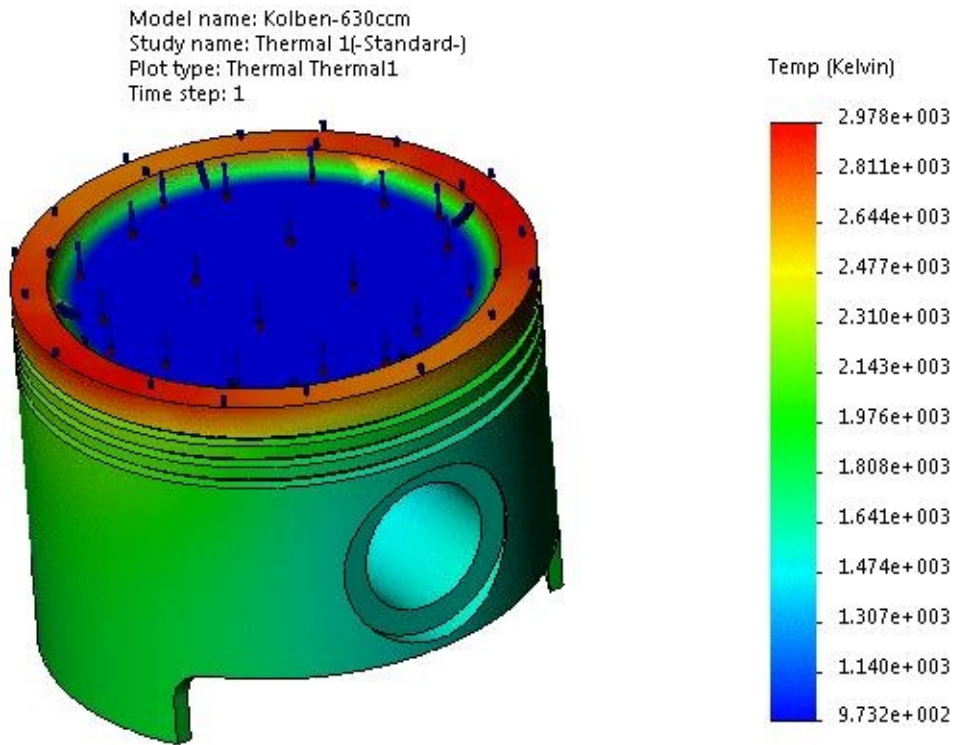


Figure-18. Solution of analysis

- To change the temperature scale, right-click on the Temperature scale at the right. A shortcut menu will be displayed; refer to [Figure-19](#).

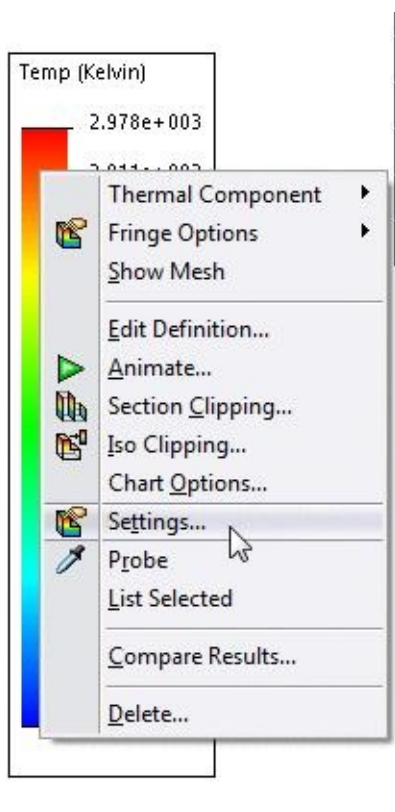


Figure-19. Shortcut menu for temperature scale

- Click on the **Settings** button from the shortcut menu. The **Thermal Plot**

PropertyManager will be displayed; refer to [Figure-20](#).

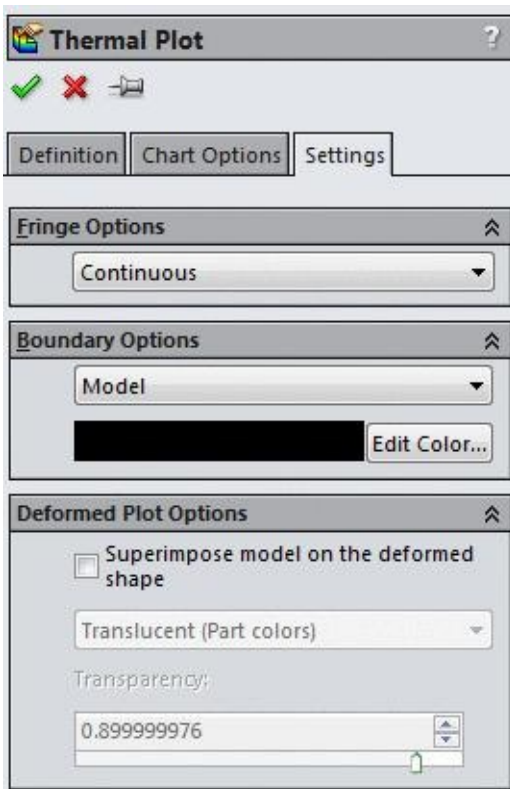


Figure-20. Thermal Plot PropertyManager

- Click on the **Definition** tab and then click on the second drop-down in the **Display** rollout; refer to [Figure-21](#).

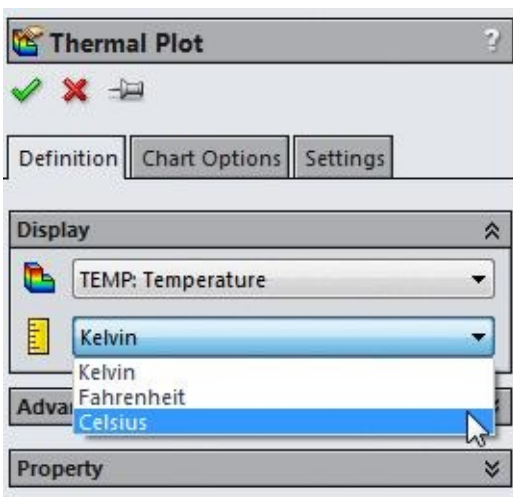


Figure-21. Definition tab of Thermal Plot PropertyManager

- Click on the **Celsius** option from the list displayed and click on the **OK** button from the **PropertyManager**. The scale will change to **Celsius**.
- You can select the other components of Thermal analysis by right-clicking on the **Temperature scale** and selecting the desired option from the Thermal Component cascading menu; refer to [Figure-22](#).

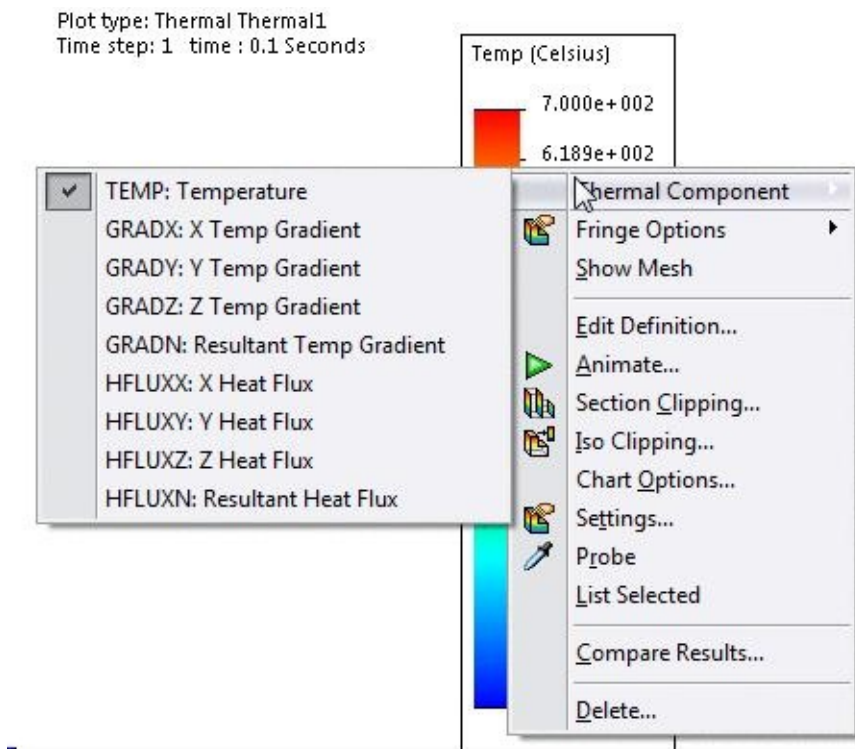


Figure-22. Thermal Components

SWITCHING FROM STEADY STATE ANALYSIS TO TRANSIENT THERMAL ANALYSIS

Earlier we have performed the steady state analysis. Now, we will use the above analysis and switch to transient thermal analysis. The procedure to switch to transient thermal analysis is given next.

- Right-click on the name of analysis in the **Analysis Manager**; refer to [Figure-23](#).
- Click on the **Properties** button from the menu displayed. The **Thermal** dialog box will be displayed; refer to [Figure-24](#).

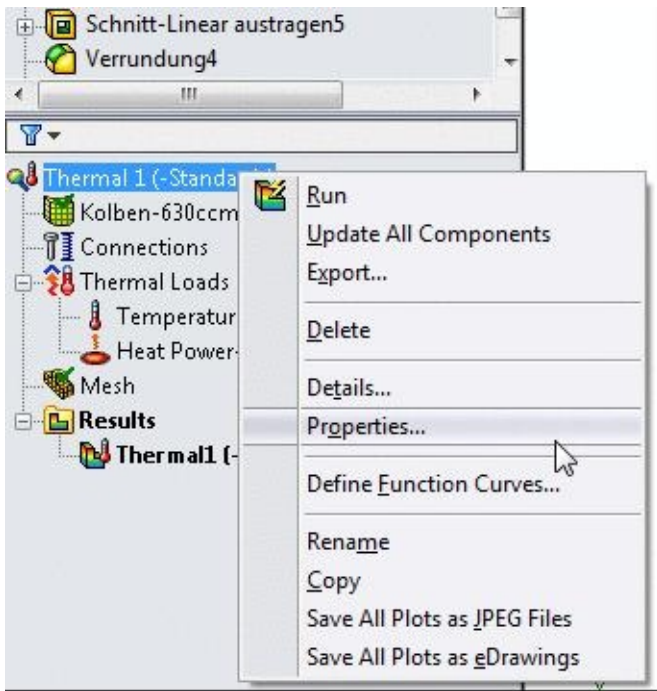
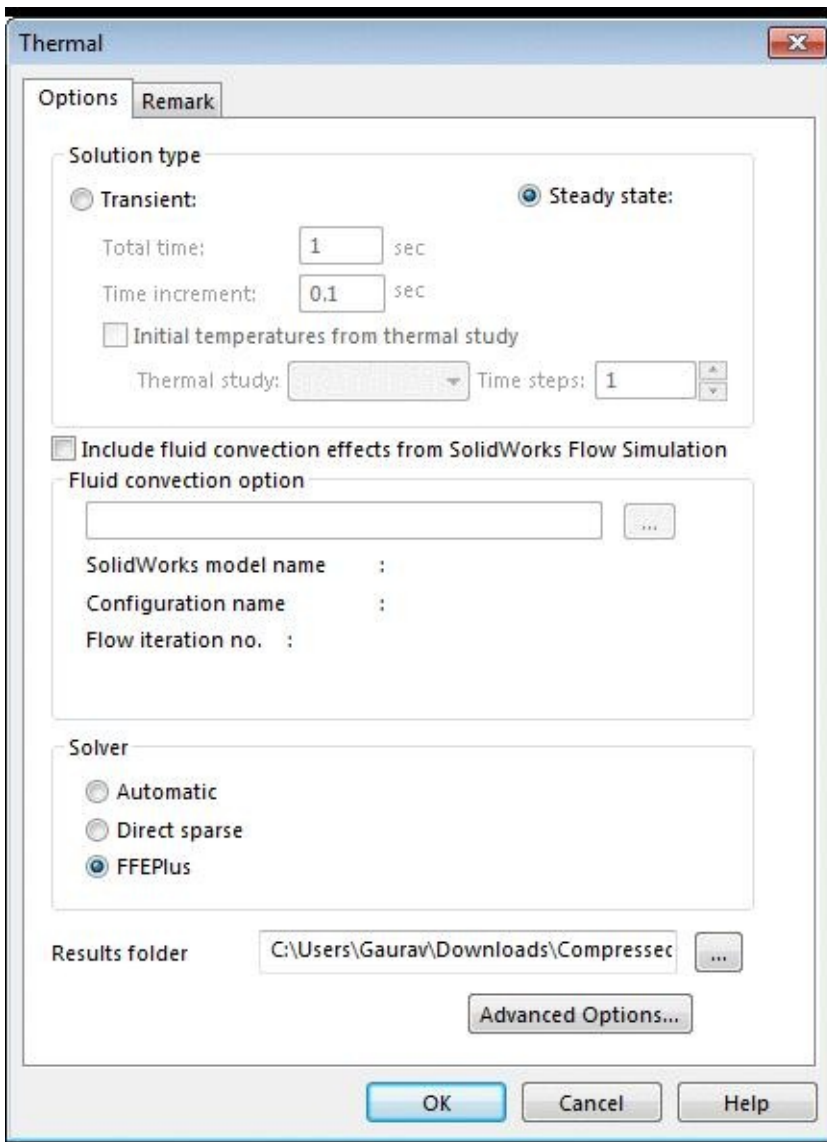


Figure-23. Shortcut menu for analysis properties

Figure-24. Thermal dialog box



- Click on the **Transient** radio button. The edit boxes below it will become active.
- Specify the total time of study (in seconds) in the **Total time** edit box.
- Click in the **Time increment** edit box and specify the increment value in seconds.
- If you have earlier performed any other analysis from where you want to import the temperature as the initial value then click on the **Initial temperature from thermal study** check box and then select the study name from the drop-down.
- You can also select the desired step number from the adjacent spinner.
- Select the desired solver from the **Solver** area of the dialog box and then click on the **OK** button from the dialog box.

Specifying Initial Temperature

Now, we need to specify the initial temperature for the analysis if we have not imported it from any earlier study.

- Click on the **Temperature** button from the **Thermal Loads** drop-down. The **Temperature PropertyManager** will be displayed.
- Select the **Initial Temperature** radio button and select the faces as shown in [Figure-25](#).

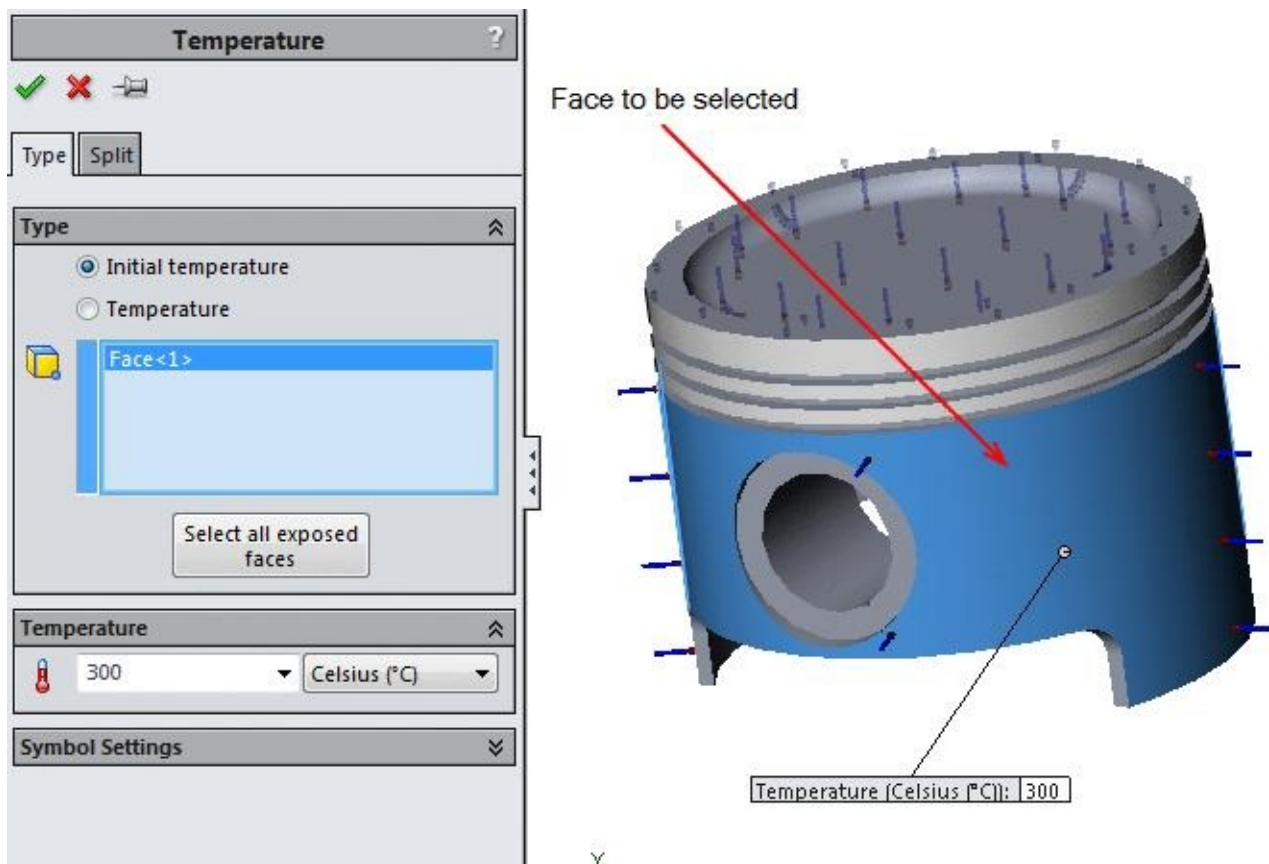


Figure-25. Faces to be selected for initial temperature

- Click in the **Temperature** edit box and in the **Temperature** rollout and specify the value

as 300 °C.

- Click on the **OK** button from the **PropertyManager** to apply the temperature.

Running the analysis

- Click on the **Run** button to run the analysis.
- After the analysis is complete, compare the two results. You will find that the analysis temperature range is changes and the part is displayed cooler than earlier one. This is because we have allowed the time for cooling in analysis and we have specified the initial temperature as 300 °C.
- When you animate the result, we will find that heat is transferred from top face to side walls as the time passes.

Using Probe to find out temperature

- After running the analysis, right-click in the modeling area. A shortcut menu will be displayed.
- Select the **Probe** button from the shortcut menu; refer to [Figure-26](#).

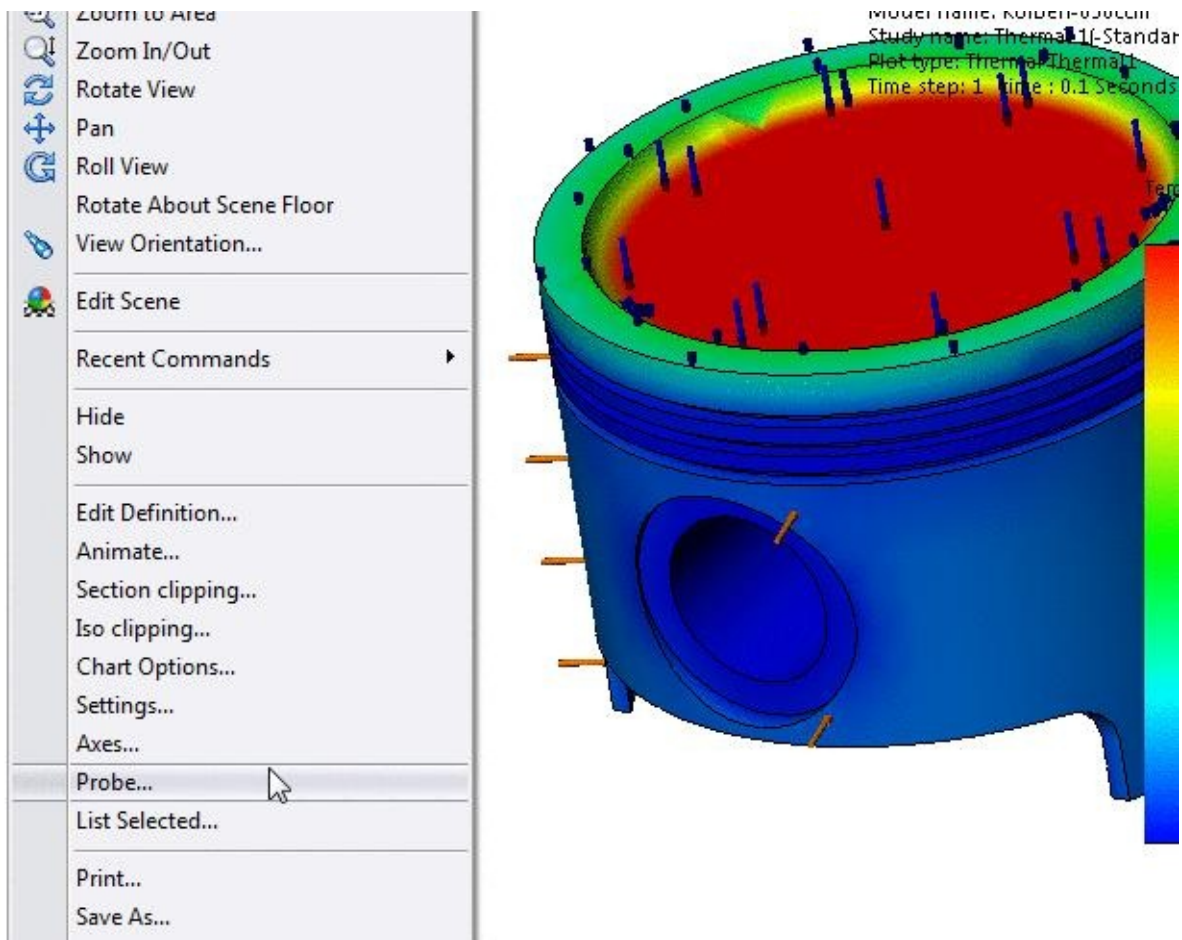


Figure-26. Probe button

- The **Probe Result PropertyManager** will be displayed; refer to [Figure-27](#).

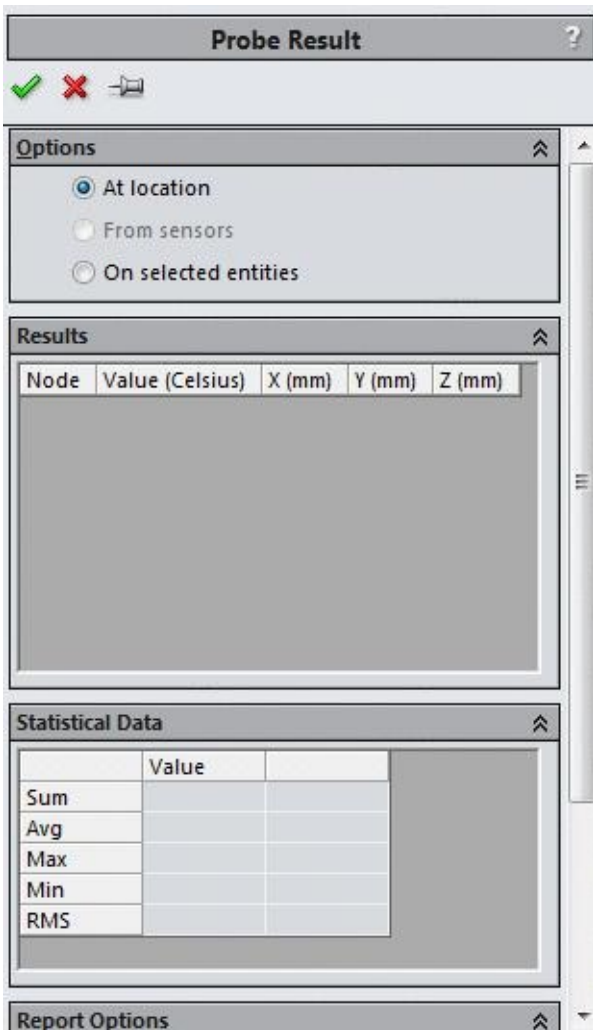


Figure-27. Probe Result PropertyManager

- Click on the model at the desired position. The temperature and other parameters of the selected position will be displayed in a box and **Results** rollout of **PropertyManager**; refer to [Figure-28](#).

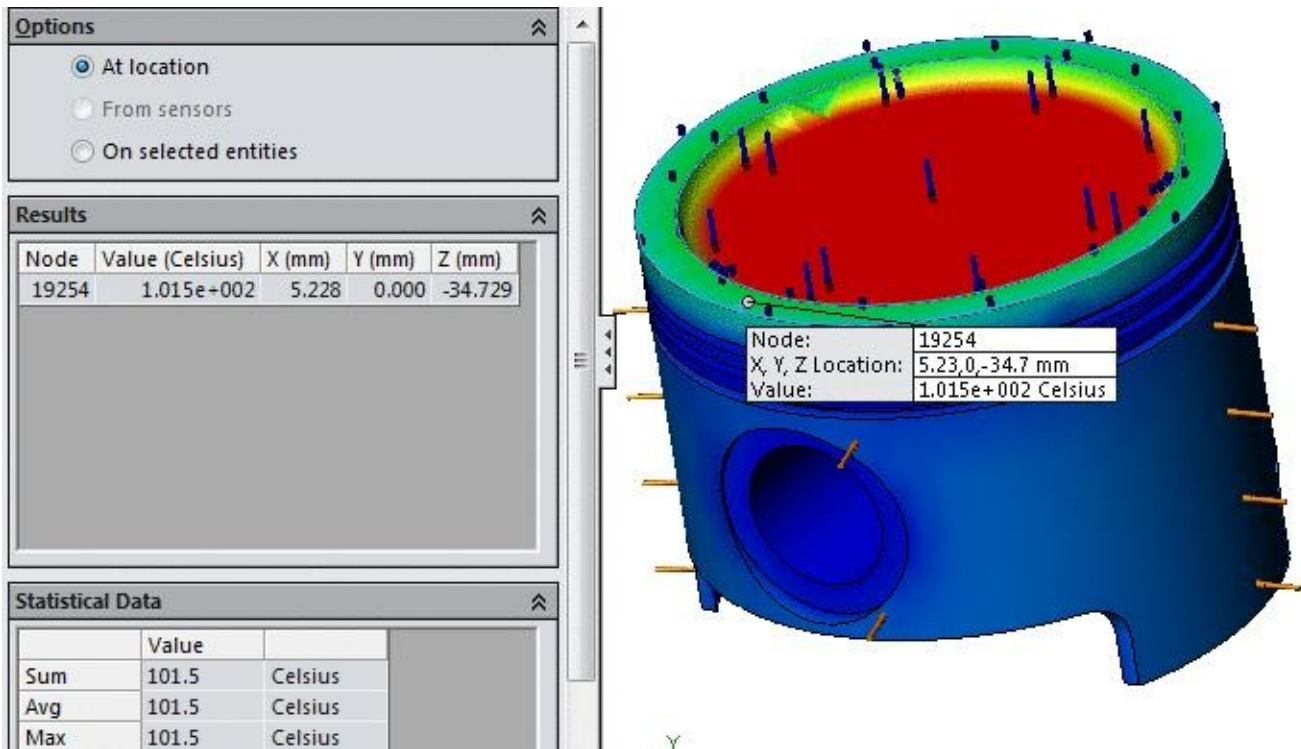


Figure-28. Temperature output

Note that to be able to perform the analysis using the SolidWorks Simulation software, you should be able to answer the following questions:

1. Heat transfer through a solid body is referred to as

- o Conduction
- o Convection
- o Radiation
- o Generation

2. The temperature gradient is defined as

- o Temperature rate of change per unit length
- o Heat flow through a body
- o Temperature rate of change per unit volume
- o Temperature rate of change per unit area

3. Heat flux is defined by

- o The amount of heat generated per unit volume
- o The heat transfer rate per unit area
- o The amount of heat stored in a control volume

o The temperature change per unit length

4. Heat transfer between a solid body and a fluid is referred to as

o Conduction o Convection o Radiation o Generation

5. Which behavior best describes a material with a high thermal conductivity compared to a material with a smaller thermal conductivity?

o Temperature change is smaller through the solid

o Heat flux is higher through the solid

o Thermal resistance is lower through the solid

o All of the above

6. The convection coefficient is related to

o The rate that heat is transferred between a solid and fluid

o The rate that heat is transferred between two solids

o The emissive power of the material

o The heat generation capability of the material

7. The amount of heat transferred by radiation is directly related to

o The Stephan-Boltzmann constant

o The surface temperature

o The emissivity of the material

o All of the above

8. If no heat is transferred to or from a surface, it is referred to as

o Isothermal o Exothermal o Adiabatic o Isobaric

9. When all temperatures are in equilibrium, the problem is assumed to be

- o Transient
- o Steady-State
- o Laminar
- o None of the above

10. Which of the following terms is not part of the energy balance for heat transfer?

- o Generated energy
- o Stored energy
- o Energy into the system
- o Energy destroyed

PROBLEM 1

A metal sphere of diameter $d = 35\text{mm}$ is initially at temperature $T_i = 700\text{ K}$. At $t=0$, the sphere is placed in a fluid environment that has properties of $T_\infty = 300\text{ K}$ and $h = 50\text{ W/m}^2\text{-K}$. The properties of the steel are $k = 35\text{ W/m-K}$, $\rho = 7500\text{ kg/m}^3$, and $c = 550\text{ J/kg-K}$. Find the surface temperature of the sphere after 500 seconds.

PROBLEM 2

A flanged pipe assembly; refer to [Figure-29](#), made of plain carbon steel is subjected to both convective and conductive boundary conditions. Fluid inside the pipe is at a temperature of 130°C and has a convection coefficient of $h_i = 160\text{ W/m}^2\text{-K}$. Air on the outside of the pipe is at 20°C and has a convection coefficient of $h_o = 70\text{ W/m}^2\text{-K}$. The right and left ends of the pipe are at temperatures of 450°C and 80°C , respectively. There is a thermal resistance between the two flanges of $0.002\text{ K-m}^2\text{/W}$. Use thermal analysis to analyze the pipe under both steady state and transient conditions.

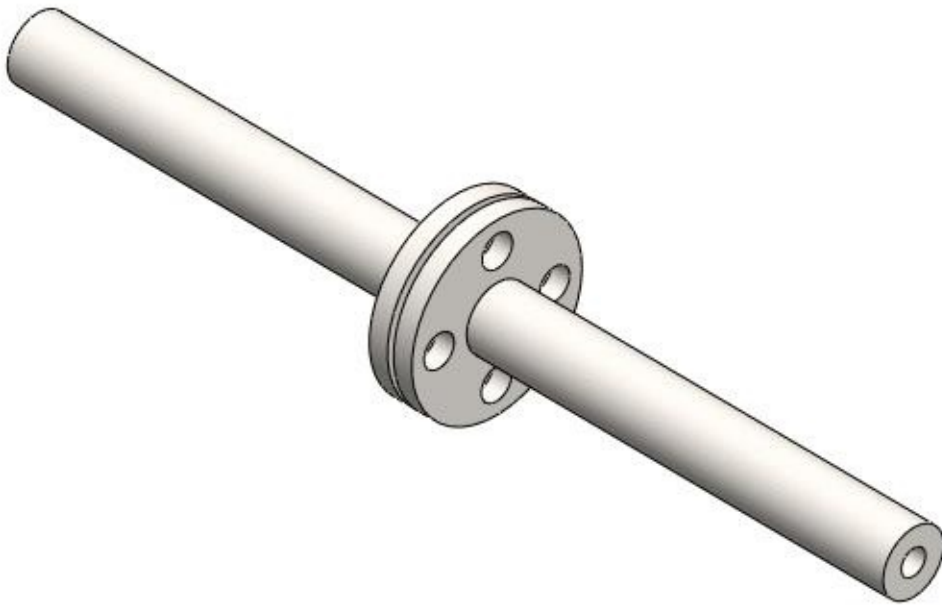


Figure-29. Flanged pipe assembly

BUCKLING ANALYSIS

CHAPTER 9



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Buckling Analysis.*
- *Applying Material.*
- *Defining Fixtures*
- *Applying loads*
- *Defining Connections*
- *Simulating Analysis.*
- *Interpreting results*

INTRODUCTION

Slender models tend to buckle under axial loading. Buckling is 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 stiffness becomes singular. The Linearized buckling approach, used here, solves an eigenvalue problem to estimate the critical buckling factors and the associated buckling mode shapes.

A model can buckle in different shapes under different levels of loading. The shape the model takes while buckling is called the buckling mode shape and the loading is called the critical or buckling load. Buckling analysis calculates a number of modes as requested in the Buckling dialog. Designers are usually interested in the lowest mode (mode 1) 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. The mode shapes can help you modify the model or the support system to prevent buckling in a certain mode.

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

In a laymen's language, if you press down on an empty soft drink can with your hand, not much will seem to happen. If you put the can on the floor and gradually increase the force by stepping down on it with your foot, at some point it will suddenly squash. This sudden scrunching is known as "buckling."

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.

In the normal use of most products, buckling can be catastrophic if it occurs. The failure is not one because of stress but geometric stability. Once the geometry of the part starts to deform, it can no longer support even a fraction of the force initially applied. The worst part about buckling for engineers is that buckling usually occurs at relatively low stress values for what the material can withstand. So they have to make a separate check to see if a product or part thereof is okay with respect to buckling.

Slender structures and structures with slender parts loaded in the axial direction buckle under relatively small axial loads. Such structures may fail in buckling while their stresses are far below critical levels. For such structures, the buckling load becomes a critical design factor. Stocky structures, on the other hand, require large loads to buckle, therefore buckling analysis is usually not required.

Buckling almost always involves compression. In civil engineering, buckling is to be avoided when designing support columns, load bearing walls and sections of bridges which may flex under load. For example an I-beam may be perfectly

“safe” when considering only the maximum stress, but fail disastrously if just one local spot of a flange should buckle! In mechanical engineering, designs involving thin parts in flexible structures like airplanes and automobiles are susceptible to buckling. Even though stress can be very low, buckling of local areas can cause the whole structure to collapse by a rapid series of ‘propagating buckling’.

Buckling analysis calculates the smallest (critical) loading required buckling 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.

USE OF BUCKLING ANALYSIS

Slender parts and assemblies with slender components 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 as failure occurs earlier due to high stresses. The procedure to use buckling analysis in SolidWorks Simulation is given next.

Starting the Buckling Analysis

- Open the part on which you want to perform the analysis.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies that can be performed, will be displayed in the left.
- Click on the **Buckling** button. The tools related to buckling analysis will be displayed in the **Ribbon**.

Applying Material

- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed; refer to [Figure-1](#).

- Select the **1060 Alloy** material under **Aluminium Alloys** node.
- Click on the **Apply** button and then select the **Close** button from the dialog box.

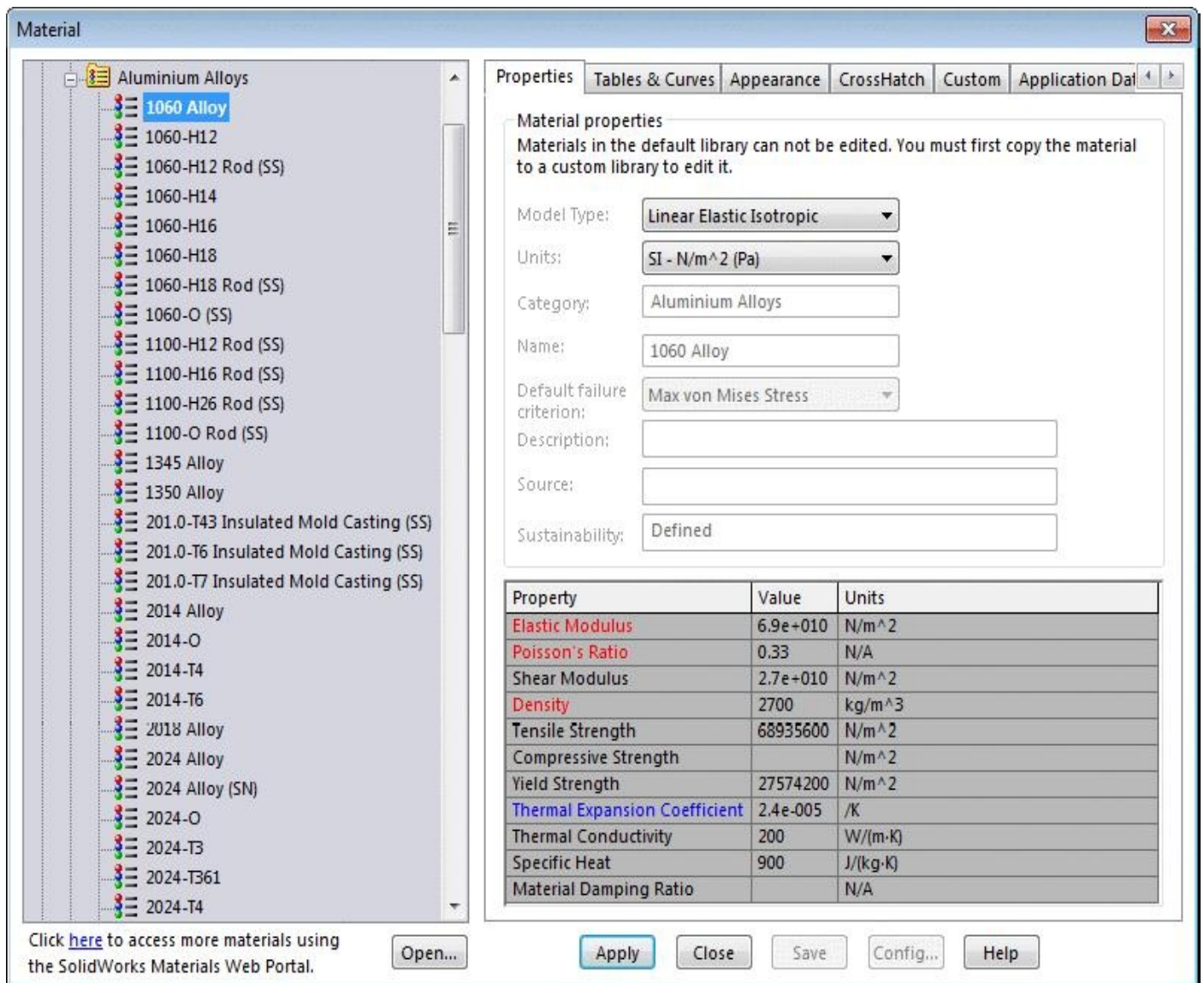


Figure-1. Material dialog box

Applying Fixture

- Click on the down arrow below **Fixtures Advisor** in the **Ribbon**. The list of tools will be displayed.
- Click on the **Fixed Geometry** button from the list. The **Fixture PropertyManager** will be displayed; refer to [Figure-2](#).

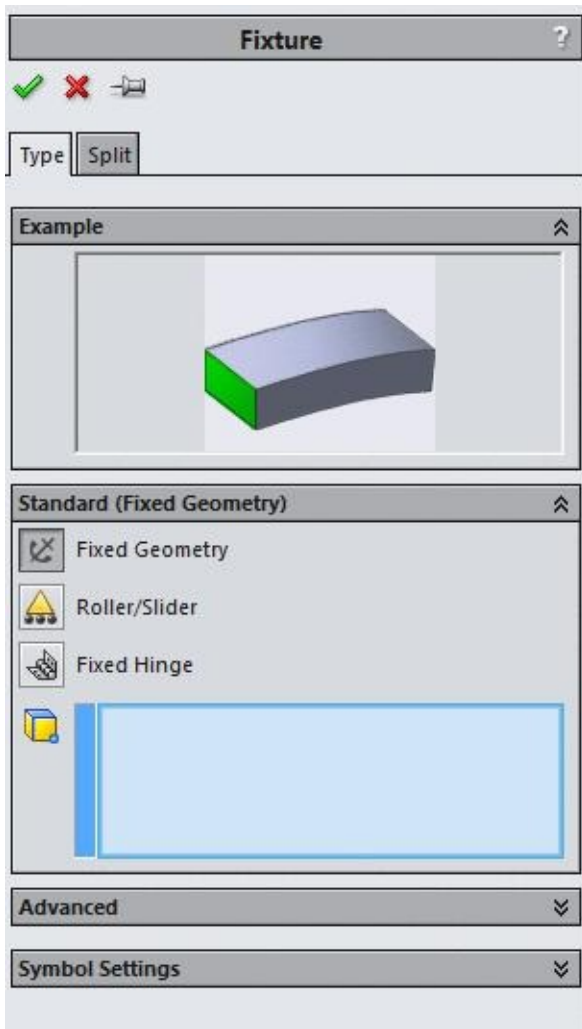


Figure-2. Fixture PropertyManager

- Select the face as shown in [Figure-3](#).



Figure-3. Face for fixture

- Click on the **OK** button from the **PropertyManager** to apply the fixture.

Applying the Force

- Click on the down arrow below **External Loads Advisor** in the **Ribbon**. A list of tools will be displayed.
- Click on the **Force** button from the list. The **Force/Torque PropertyManager** will be displayed; refer to [Figure-4](#).

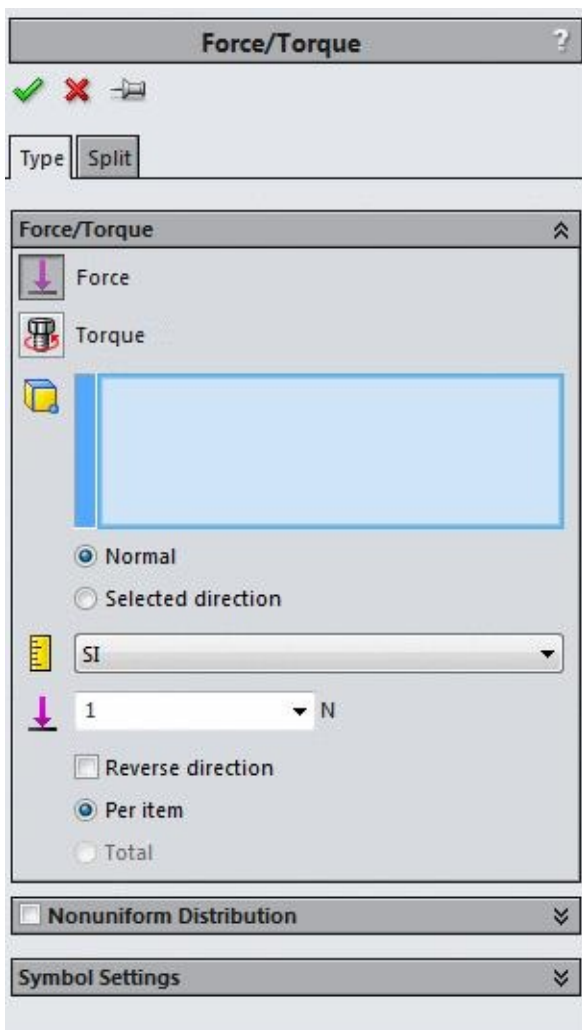


Figure-4. Force Torque PropertyManager

- Select the face as shown in [Figure-5](#).
- Click in the **Force Value** edit box in the **PropertyManager** and specify the value of force as **1000**.

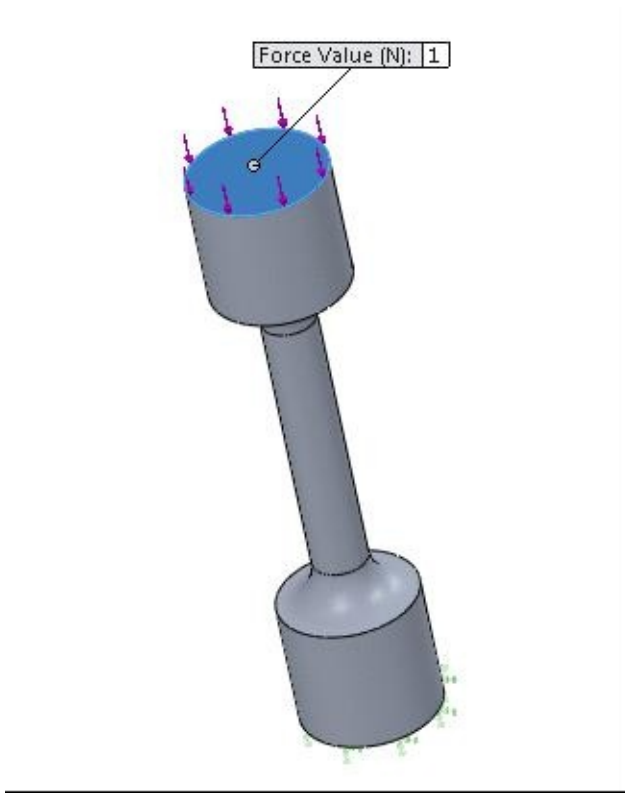


Figure-5. Face for applying force

- Click on the **OK** button from the **PropertyManager** to apply the forces.

Running the Analysis

- Click on the **Run** button from the **Ribbon**. The analysis will start solving.
- After the analyzing is complete, the result will be displayed in the modeling area; refer to [Figure-6](#).

Model name: Part1
Study name: Buckling 1(-Default-)
Plot type: Buckling Displacement1
Mode Shape : 1 Load Factor = 2.7792
Deformation scale: 17.5633

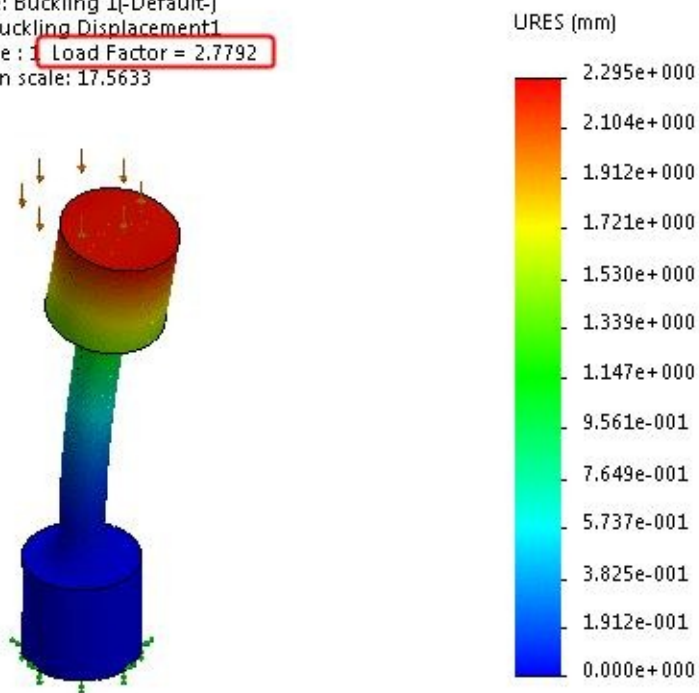


Figure-6. Result

- From the above figure, you can check that the load factor for this model is **2.7792**. This load factor represents the factor of safety for the model.
- According to the above value, the model is able to sustain the load for buckling.
- Note that although the part will not fail under the given load but it will be deformed as per the scale.

FATIGUE ANALYSIS

CHAPTER 10



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Fatigue Analysis.*
- *Selecting the previous analysis*
- *Defining the number of cycles*
- *Simulating Analysis.*
- *Interpreting results*

INTRODUCTION

Repeated loading and unloading weakens objects over time even when the induced stresses are considerably less than the allowable stress limits. This phenomenon is known as fatigue. Each cycle of stress fluctuation weakens the object to some extent. After a

number of cycles, the object becomes so weak that it fails. Fatigue is the prime cause of the failure of many objects, especially those made of metals. Examples of failure due to fatigue include, rotating machinery, bolts, airplane wings, consumer products, offshore platforms, ships, vehicle axles, bridges, and bones.

Linear and nonlinear structural studies do not predict failure due to fatigue. They calculate the response of a design subjected to a specified environment of restraints and loads. If the analysis assumptions are observed and the calculated stresses are within the allowable limits, they conclude that the design is safe in this environment regardless of how many times the load is applied.

Results of static, nonlinear, or time history linear dynamic studies can be used as the basis for defining a fatigue study. The number of cycles required for fatigue failure to occur at a location depends on the material and the stress fluctuations. This information, for a certain material, is provided by a curve called the SN curve.

Stages of Failure Due to Fatigue

Failure due to fatigue occurs in three stages:

Stage 1 One or more cracks develop in the material. Cracks can develop anywhere in the material but usually occur on the boundary faces due to higher stress fluctuations. Cracks can occur due to many reasons. Imperfections in the microscopic structure of the materials and surface scratches caused by tooling or handling are some of them.

Stage 2 Some or all the cracks grow as a result of continued loading.

Stage 3 The ability of the design to withstand the applied loads continue to deteriorate until failure occurs.

Fatigue cracks start on the surface of a material. Strengthening the surfaces of the model increases the life of the model under fatigue events.

From the above theory, you can find out that the main player in Fatigue analysis is the properties of materials rather the S-N curves.

The procedure to perform the Fatigue analysis is given next.

Starting Fatigue Analysis

- Open a part on which you have already performed Static or dynamic analysis. (We have opened the file used in Chapter 3 of this book; refer to [Figure-1](#).)

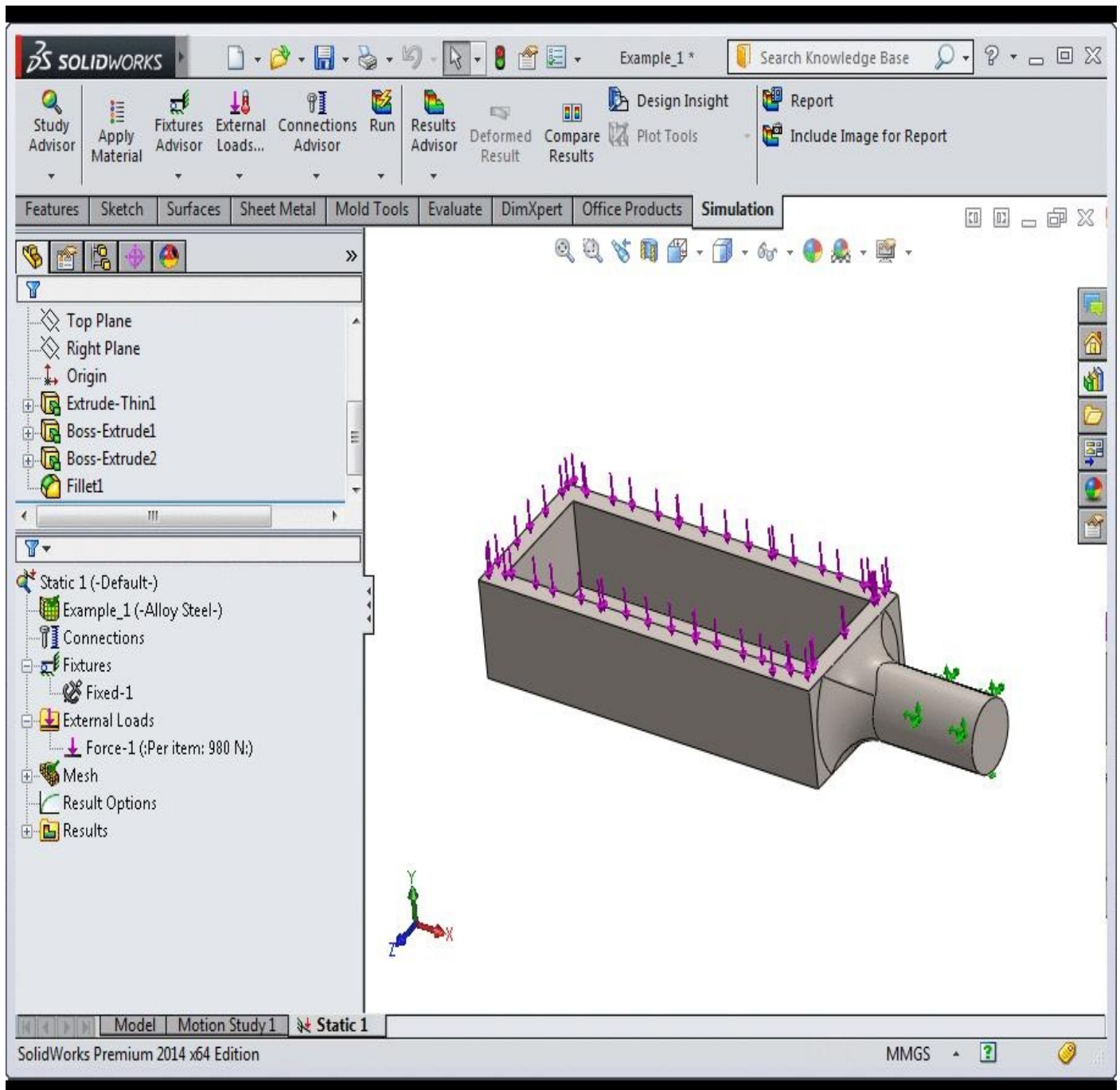


Figure-1. Model of chapter 3

- Click on the down arrow below **Study Advisor**, the list of tools will be displayed.
- Click on the **New Study** button from the list. The **Study PropertyManager** will be displayed.
- Click on the **Fatigue** button from the list and then click on the **Variable amplitude history data** button from the **Options** rollout of the **PropertyManager**; refer to [Figure-2](#).

Note that in this example, we will study the Variable amplitude history data fatigue study but in SolidWorks 2015, we can perform four type of fatigue analyses viz. **Constant amplitude events with defined cycles**, **Variable amplitude history data**, **Harmonic-fatigue of sinusoidal loading**, and **Random vibration-fatigue of random vibration**. The procedure of all the studies is almost the same.

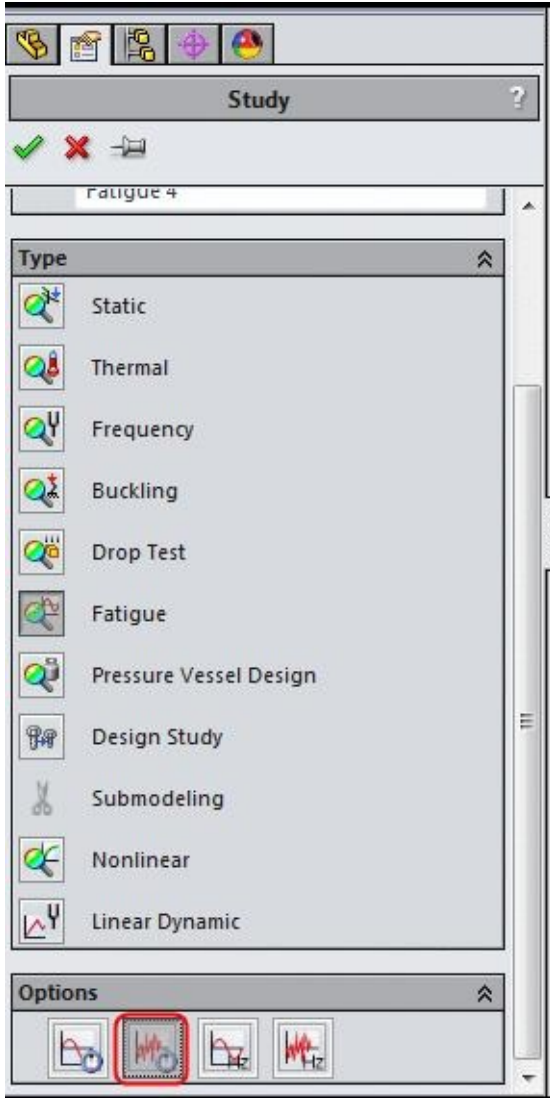


Figure-2. Variable amplitude history data button

In case of **Constant amplitude events with defined cycles**, the load being applied on the object is constant and we specify the number of loading cycles to test the fatigue.

In case of **Variable amplitude history data**, the load applied on object varies with time. This variation can be defined in the fatigue study itself.

In case of **Harmonic- fatigue of sinusoidal loading**, the dynamic load is applied in the form of sinusoidal curve. For this type of fatigue study, we require the data from studies performed under Linear Dynamic category.

In case of **Random vibration-fatigue of random vibration**, we require the data from random vibration study. To perform this study, you need to perform the random vibration study first.

- Note that if you want to study for a constant amplitude that you can select the **Constant amplitude events with defined cycles** button from the **Options** rollout.
- Click on the **OK** button from the **PropertyManager** to start the analysis. The tools related to fatigue analysis will be displayed; refer to [Figure-3](#).

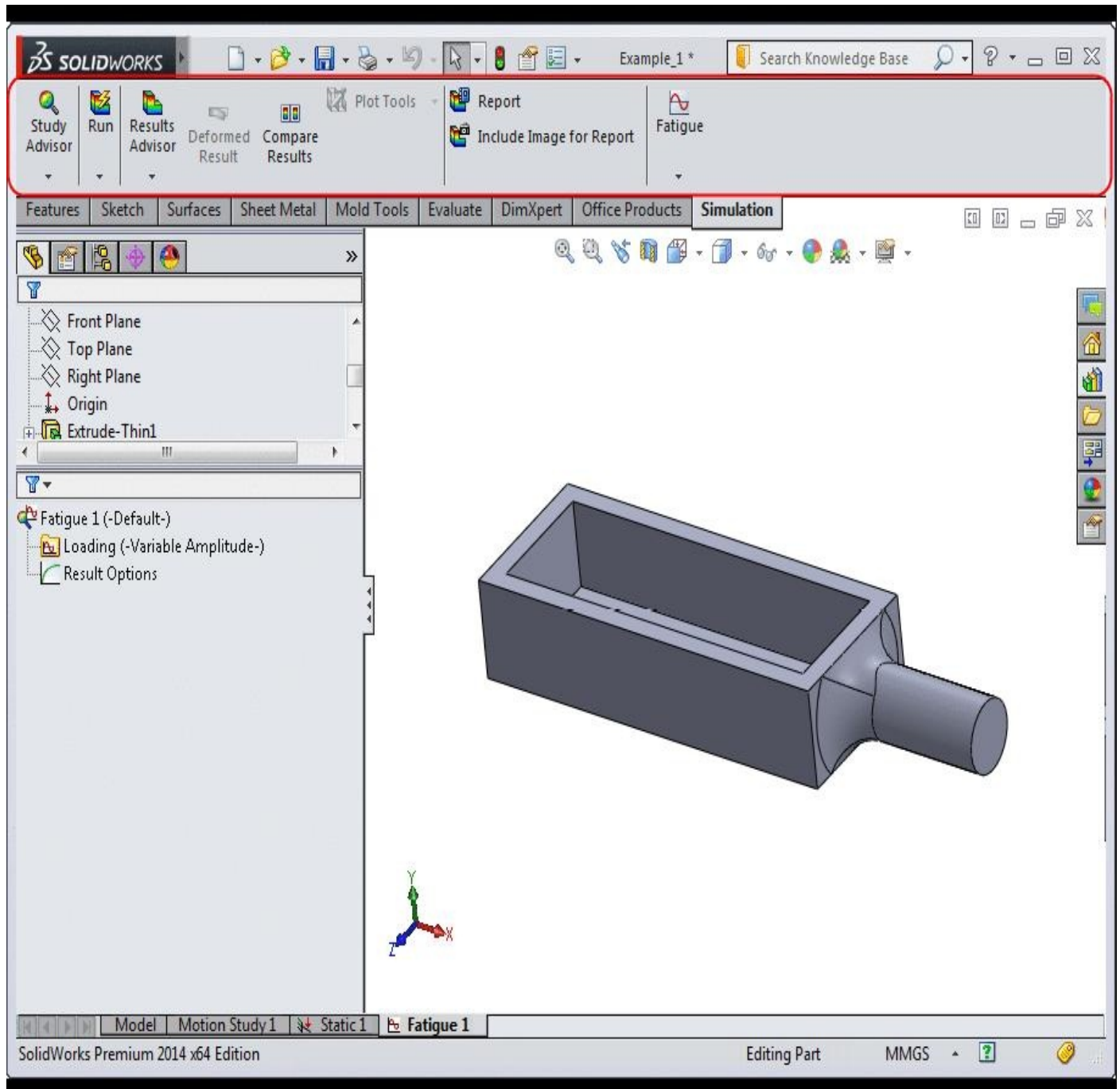


Figure-3. Tools for fatigue analysis

Initiating Fatigue Analysis

- Click on the down arrow below **Fatigue** in the **Ribbon**. The **Add Event** tool will be displayed.
- Click on the **Add Event** tool. The **Add Event (Variable) PropertyManager** will be displayed; refer to [Figure-4](#).

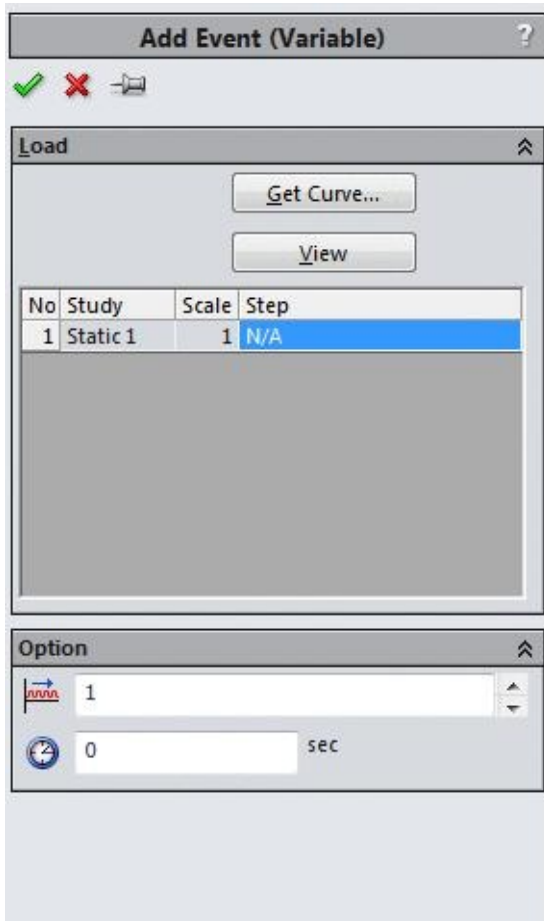


Figure-4. Add Event PropertyManager

- By default, Static1 study is added in the event list.
- Click in the **Number of repeats** spinner in the **Options** rollout and specify the number of repetitions as **300** for the load curve being generated in the next segment.

Now, we need to specify the time curve for the fatigue analysis.

- Click on the **Get Curve** button from the **PropertyManager**. The **Load History Curve** dialog box will be displayed; refer to [Figure-5](#).

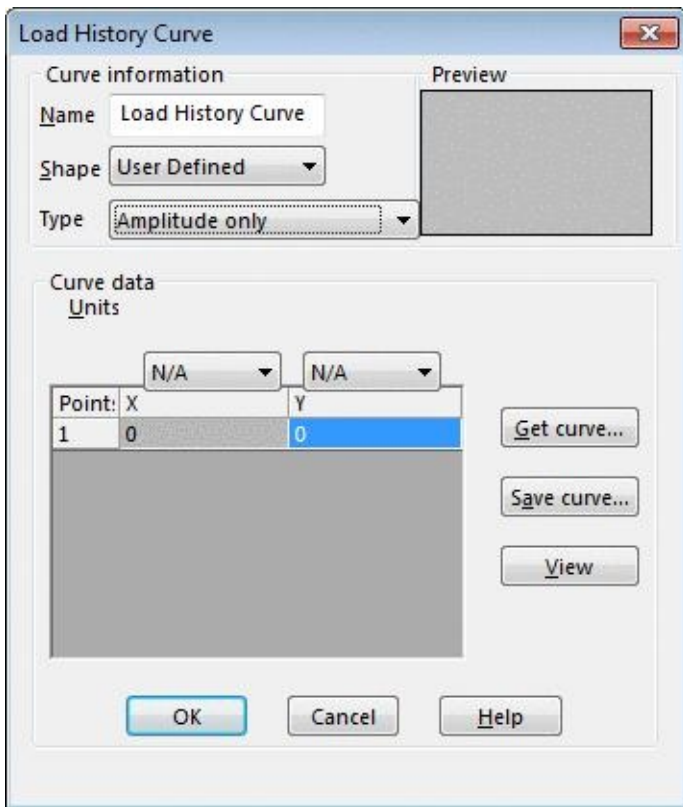


Figure-5. Load History Curve dialog box

- Click in the **Name** field and specify the name of the curve you want to create.
- Click on the **Type** drop-down and select the **Time & amplitude** option from the list displayed.
- The table of time versus amplitude is displayed in the **Curve data** area.
- Double-click on the **1** button under the **Point** column. The next point will be added in the table; refer to [Figure-6](#).

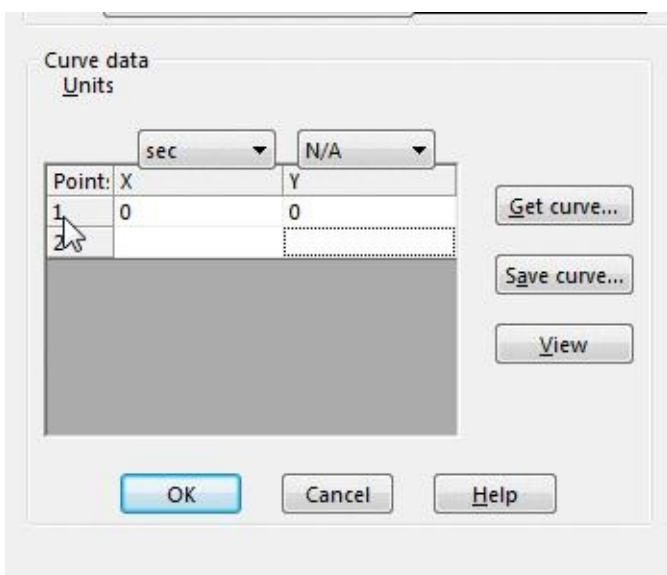


Figure-6. Curve data after adding one more point

- Similarly, you can add more points.

- Enter the data related to amplitude in the table; refer to [Figure-7](#).

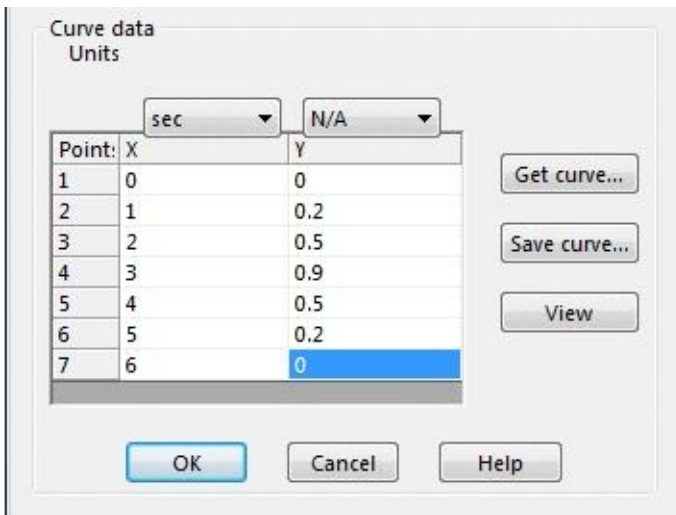


Figure-7. Curve data table with custom values

- Click on the **OK** button from the dialog box.
- Click on the **OK** button from the **PropertyManager** to add the event.

Applying or Editing S-N data

- Right-click on the name of the model in the **Analysis Manager** and select the **Apply/Edit Fatigue data** button; refer to [Figure-8](#).

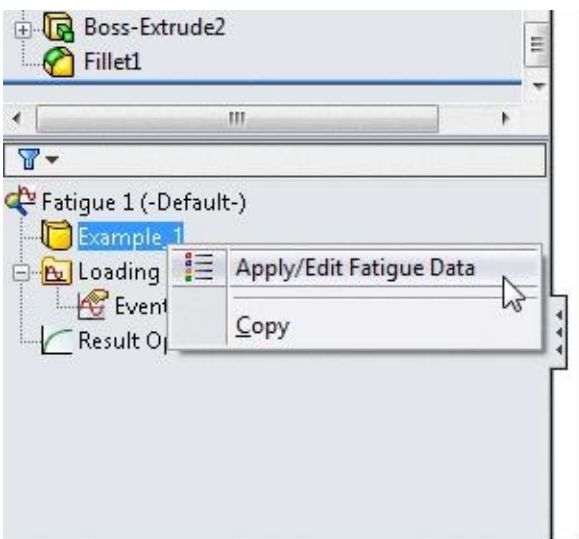


Figure-8. Apply Edit Fatigue Data button

- The **Fatigue SN Curves** tab in **Materials** dialog box will be displayed; refer to [Figure-9](#).

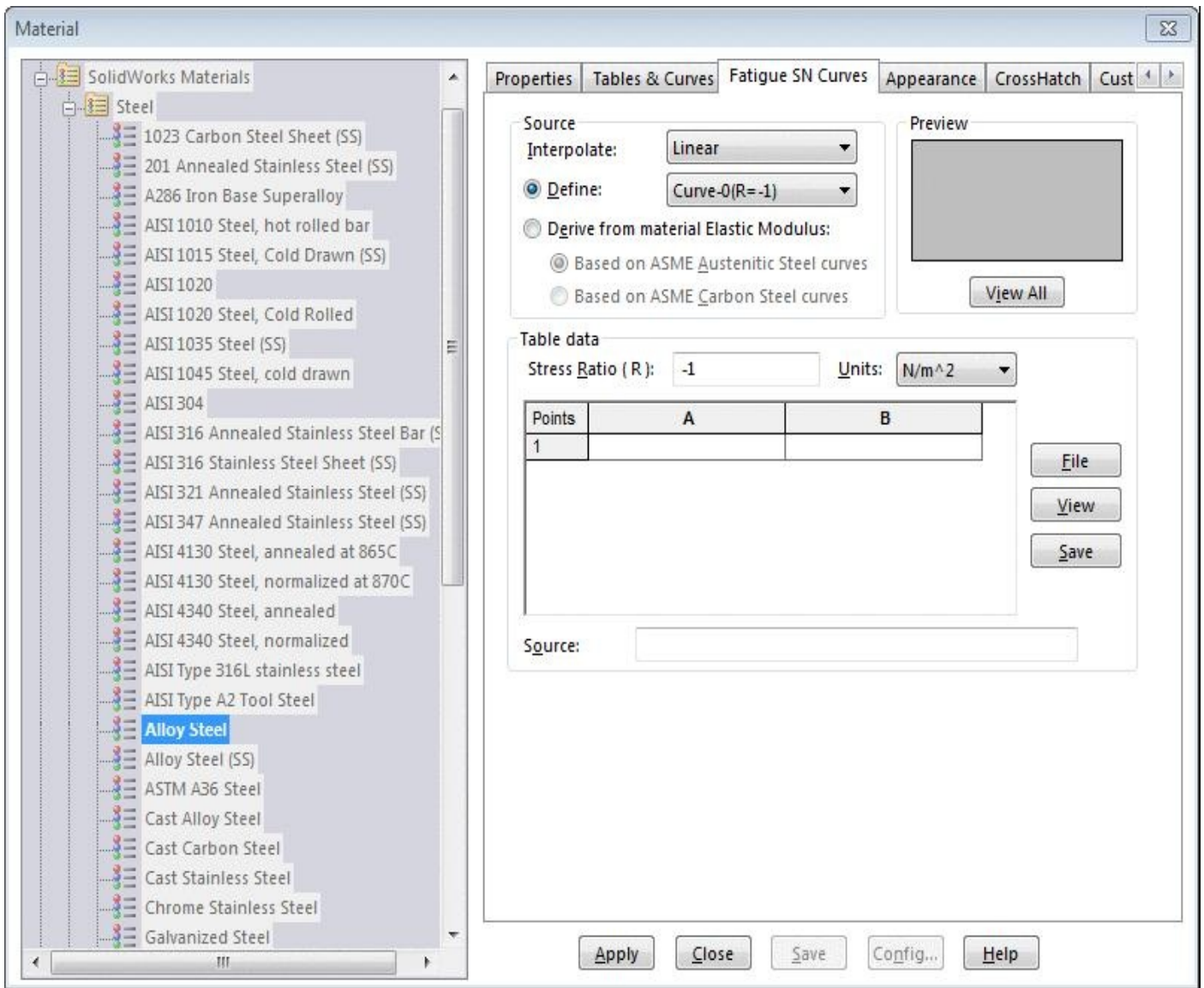


Figure-9. Fatigue SN Curve tab in Materials dialog box

- If you have the table of parameters then specify the data as done in case of **Curve Data**. In this case, we will use the fatigue data of **ASME Austenitic Steel**.
- Click on the **Derive from material Elastic Modulus** radio button from the **Source** area in the dialog box. The data will be imported automatically in the table.
- Click on the **Apply** button and then **Close** button from the dialog box.

Modifying the Static Analysis earlier performed

- Click on the **Static 1** tab in the bottom bar of **SolidWorks**; refer to [Figure-10](#). The static analysis will open; refer to [Figure-11](#).

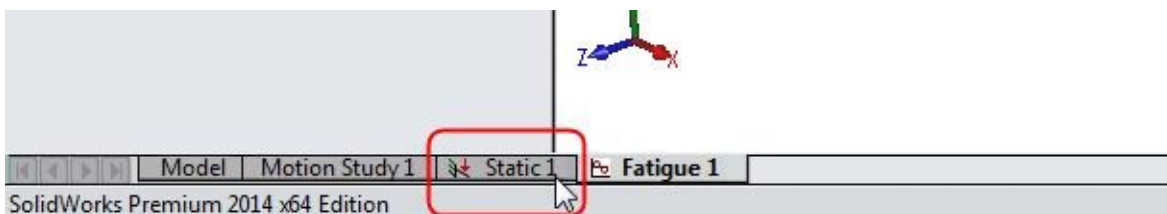


Figure-10. Static 1 tab

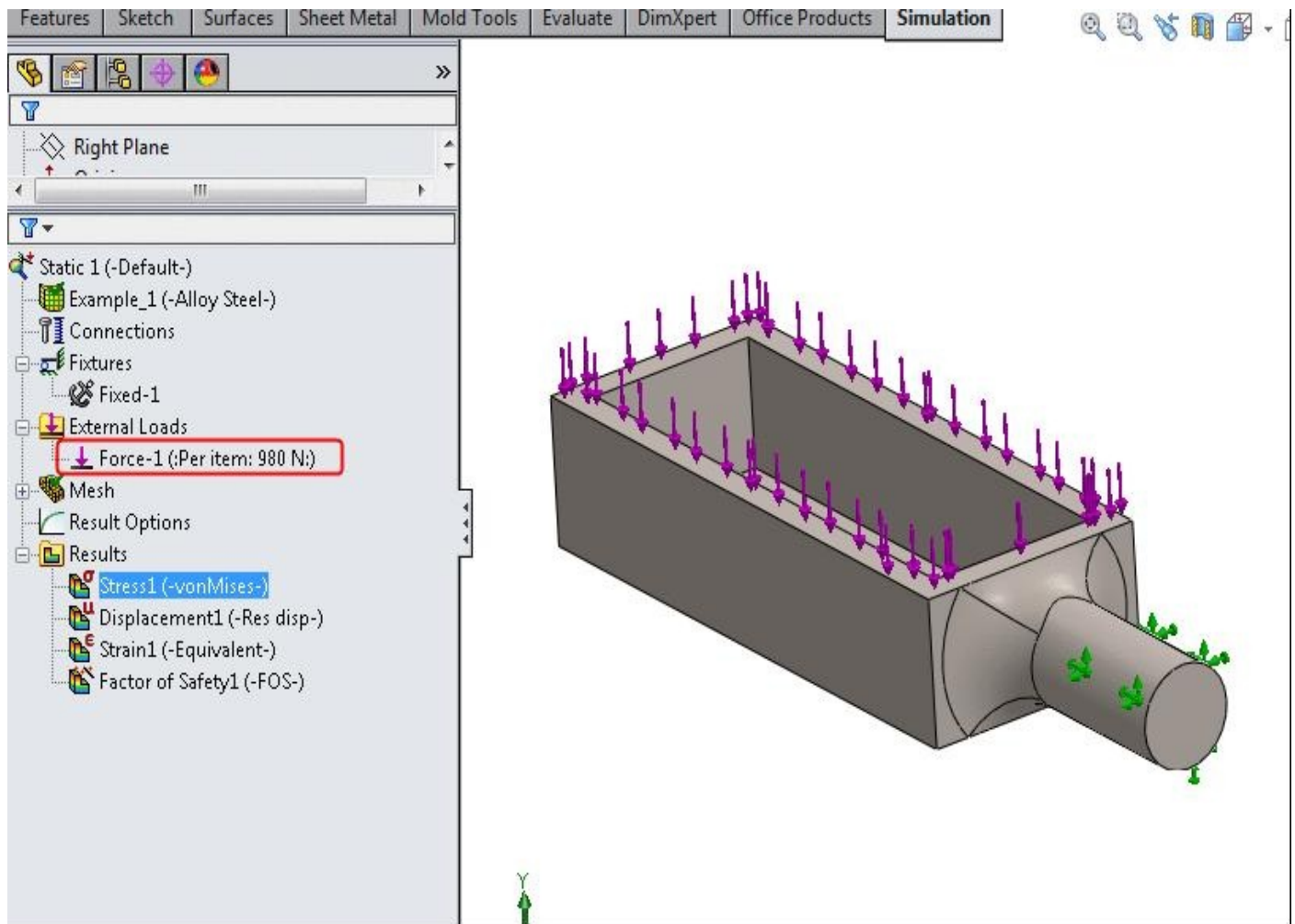


Figure-11. Static analysis earlier performed

- Double click on the **Force-1** node in the **Analysis Manager**; refer to [Figure-11](#). The **Force/Torque PropertyManager** will be displayed.
- Change the force from the **PropertyManager** to **1800** and click on the **OK** button from the **PropertyManager**.
- Click on the **Run** button from the **Ribbon** to update the analysis.

Running the Fatigue Analysis

- Click on the **Fatigue 1** tab from the bottom bar of SolidWorks. You will switch back to fatigue analysis.
- Click on the **Run** button from the **Ribbon**.
- Double-click on the **Results2 (-Life-)** node in the **Analysis Manager**. The results will be displayed as shown in [Figure-12](#).

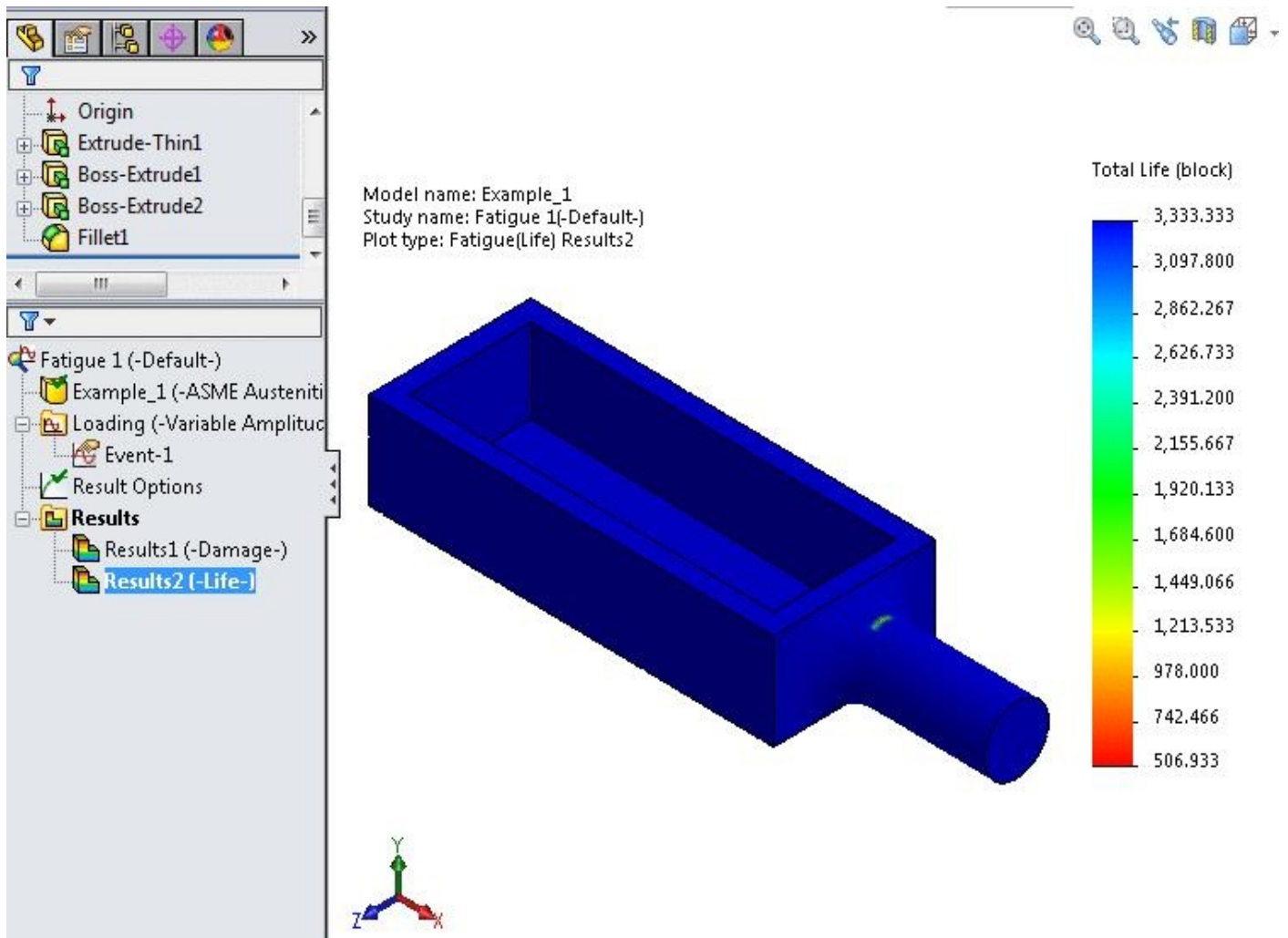


Figure-12. Results

- From the results, we can find out that the part will fail on **506th** cycle at the location painted in red.
- The red location is below the green marked area, so you need to rotate the model; refer to [Figure-13](#).

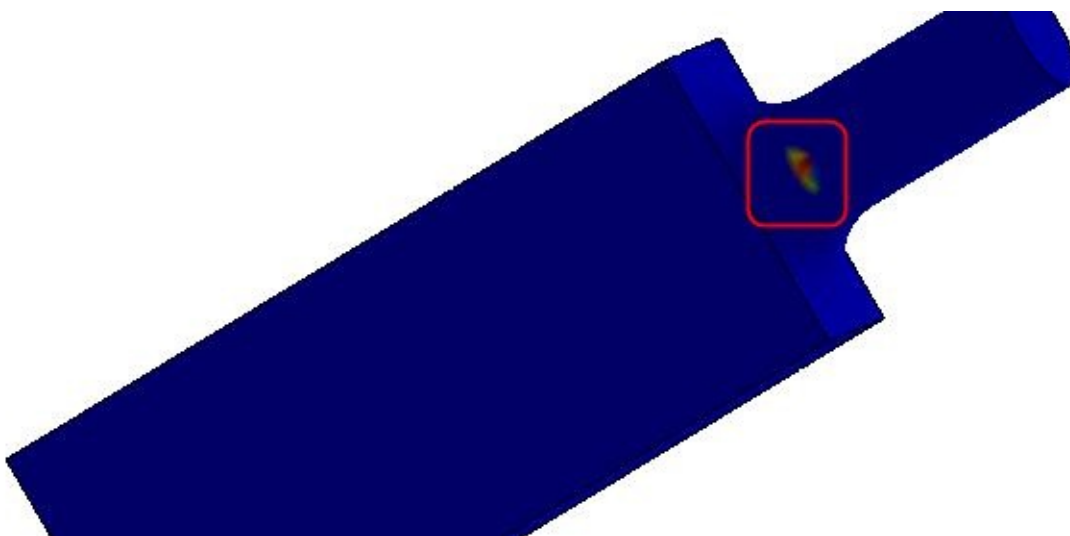


Figure-13. Location in red

Changing Properties of Fatigue Analysis

- Right-click on the **Fatigue 1 (-Default-)** node in the **Analysis Manager**. The options related to fatigue analysis will be displayed.
- Click on the **Properties** button from the list displayed. The **Fatigue** dialog box will be displayed; refer to [Figure-14](#).

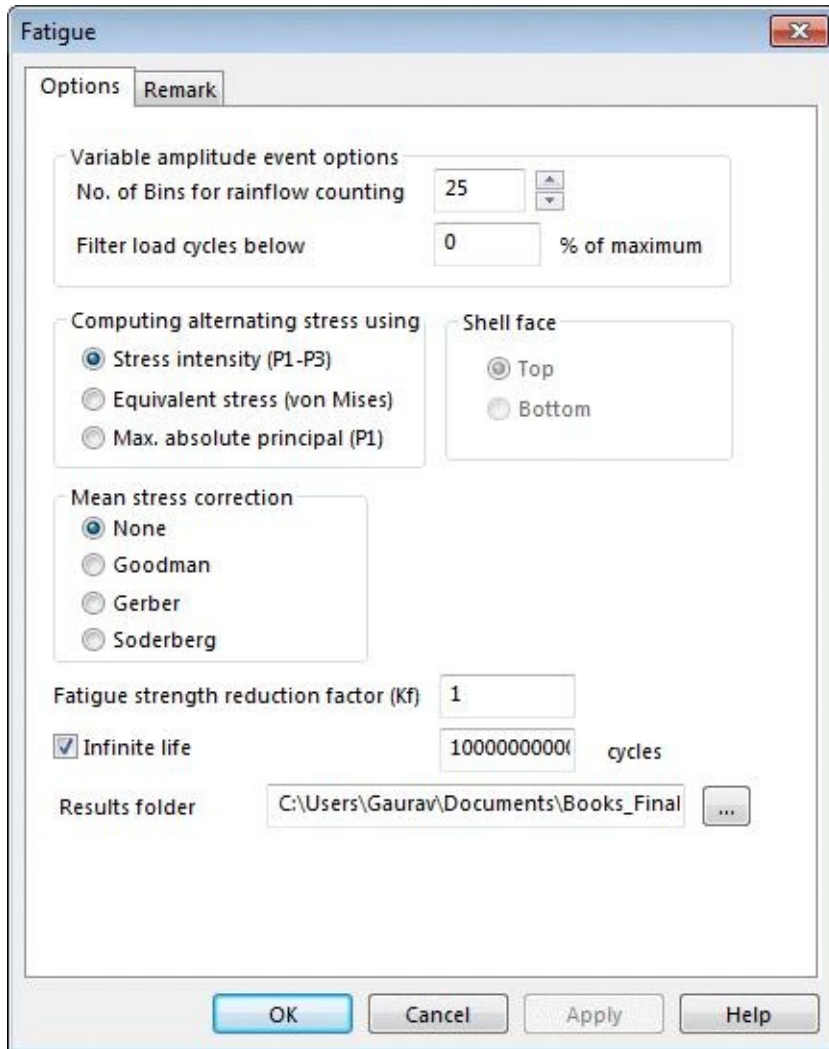


Figure-14. Fatigue dialog box

- The **No. of Bins for rainflow counting** spinner is used to specify the number of divisions of load frequency for specified amplitude.
- The options in the **Computing alternating stress using** area are used to specify the analysis result to be used for testing fatigue.
- The options in the **Mean stress correction** area are used to set the correction on the basis of material properties. **Described as:**

None: No correction.

Goodman method: Generally suitable for brittle materials.

Gerber method: Generally suitable for ductile materials.

Soderberg method: Generally the most conservative.

- When meshing is done, SolidWorks creates two type of shell elements: Top face shell elements and Bottom face shell elements. This elements can be distinguished by their color. The Shell face area of the dialog box allows you to select the faces that you want to use for performing fatigue analysis.
- The **Fatigue strength reduction factor (Kf)** edit box is used to specify the value of fatigue reduction factor for allowing environment factor in analysis. Use this factor, between 0 and 1, to account for differences in test environment used to generate the S-N curve and the actual loading environment. The program divides the alternating stress by this factor before reading the corresponding number of cycles from the S-N curve. This is equivalent to reducing the number of cycles that cause failure at a certain alternating stress. Fatigue handbooks suggest numeric values for the fatigue strength reduction factor.
- The Filter load cycles below edit box forces the system to filter out load cycles with ranges smaller than the specified percentage of the maximum range. For example, if you specify 5%, the program ignores cycles with load ranges less than 5% of the maximum range of the load history. Use this parameter to filter out noise from measuring devices.

CHAPTER 11



Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Starting Drop-Test*
- *Applying Material*
- *Setting drop parameters*
- *Simulating Analysis.*
- *Interpreting results*

INTRODUCTION

In a drop test analysis, the time varying stresses and deformations due to an initial impact of the product with a rigid or flexible planar surface (the floor) are calculated. As the product deforms, secondary internal and external impacts are also calculated, locating

critical weaknesses or failure points, as well as stresses and displacements. Drop test analysis using SolidWorks Simulation enables you to visualize the elastic stress wave propagating through the system so that the correct assembly methods are used.

The maximum “g force” experienced by individual components is one of the primary unknowns before a drop test. This is a critical parameter since many electronic and mechanical components are not rated for further use above a specified maximum g force. Using drop test analysis with SolidWorks Simulation, designers and engineers can measure the time varying accelerations (g force) at any location within the product, providing critical design information and reducing the number of physical tests needed. A design team can easily verify performance while they design and select the correct materials, component shape, and fixture methods to ensure that critical components stay within their “max g force” limits.

STARTING DROP TEST

- Open the part on which you want to perform the drop-test.
- Click on the **SolidWorks Simulation** button from the **SOLIDWORKS Add-Ins** tab of the **Ribbon** to add **Simulation** tab in the **Ribbon**, if not added already.
- Click on the Down arrow below the **Study Advisor** button and select the **New Study** tool from the drop-down. List of analysis studies will be displayed in the **Study PropertyManager**.
- Click on the **Drop Test** button from the list and click on the **OK** button from the **PropertyManager**. The tools related to drop-test will be displayed; refer to [Figure-1](#).

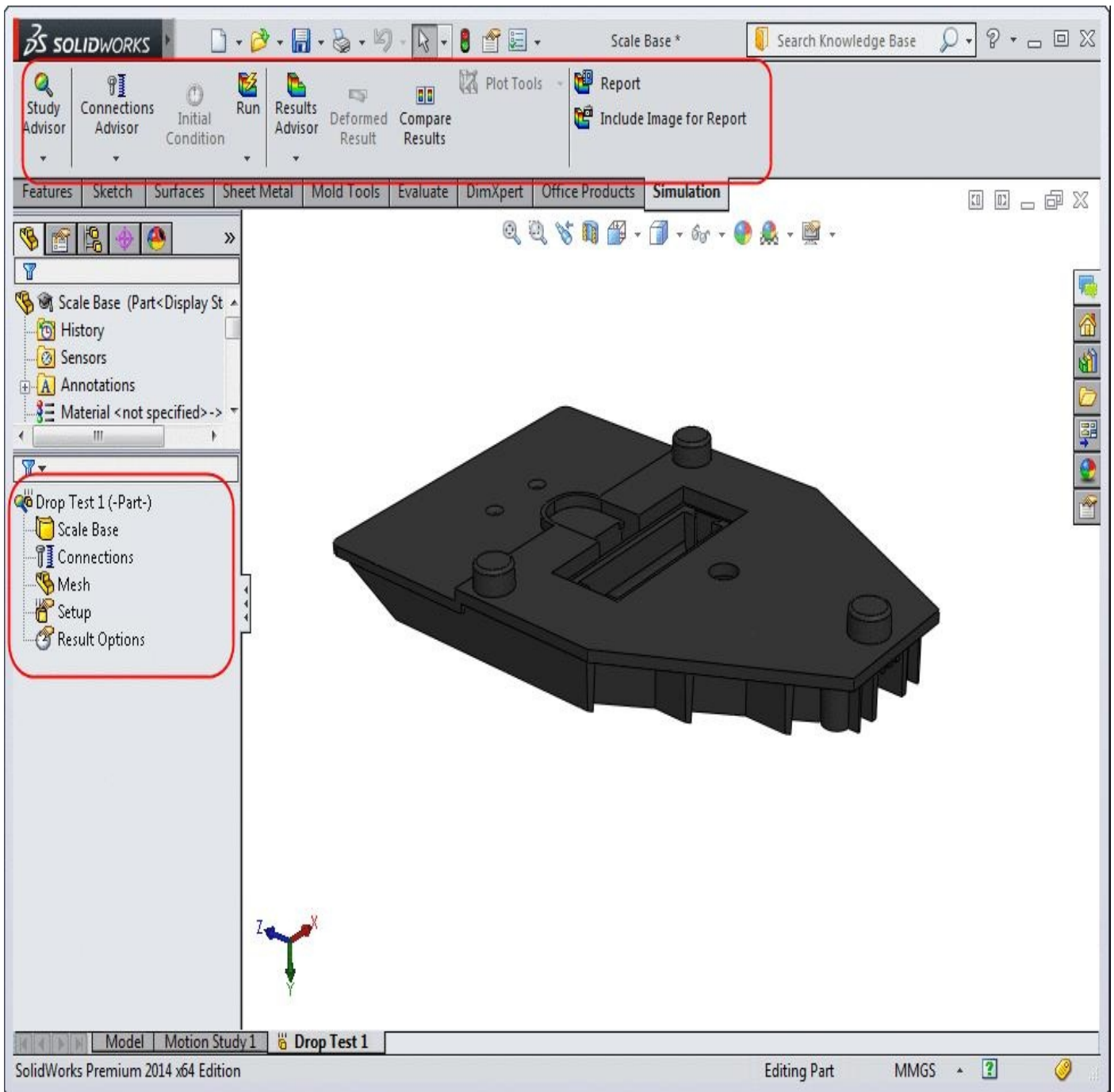


Figure-1. Tools related to drop test

Applying Material

- Right-click on name of the model i.e. Scale Base. A shortcut menu will be displayed.
- Click on **Apply/Edit Material** button from the shortcut menu. The **Material** dialog box will be displayed.
- Select the **ABS PC Material** from the **Plastic** category in the left; refer to [Figure-2](#).

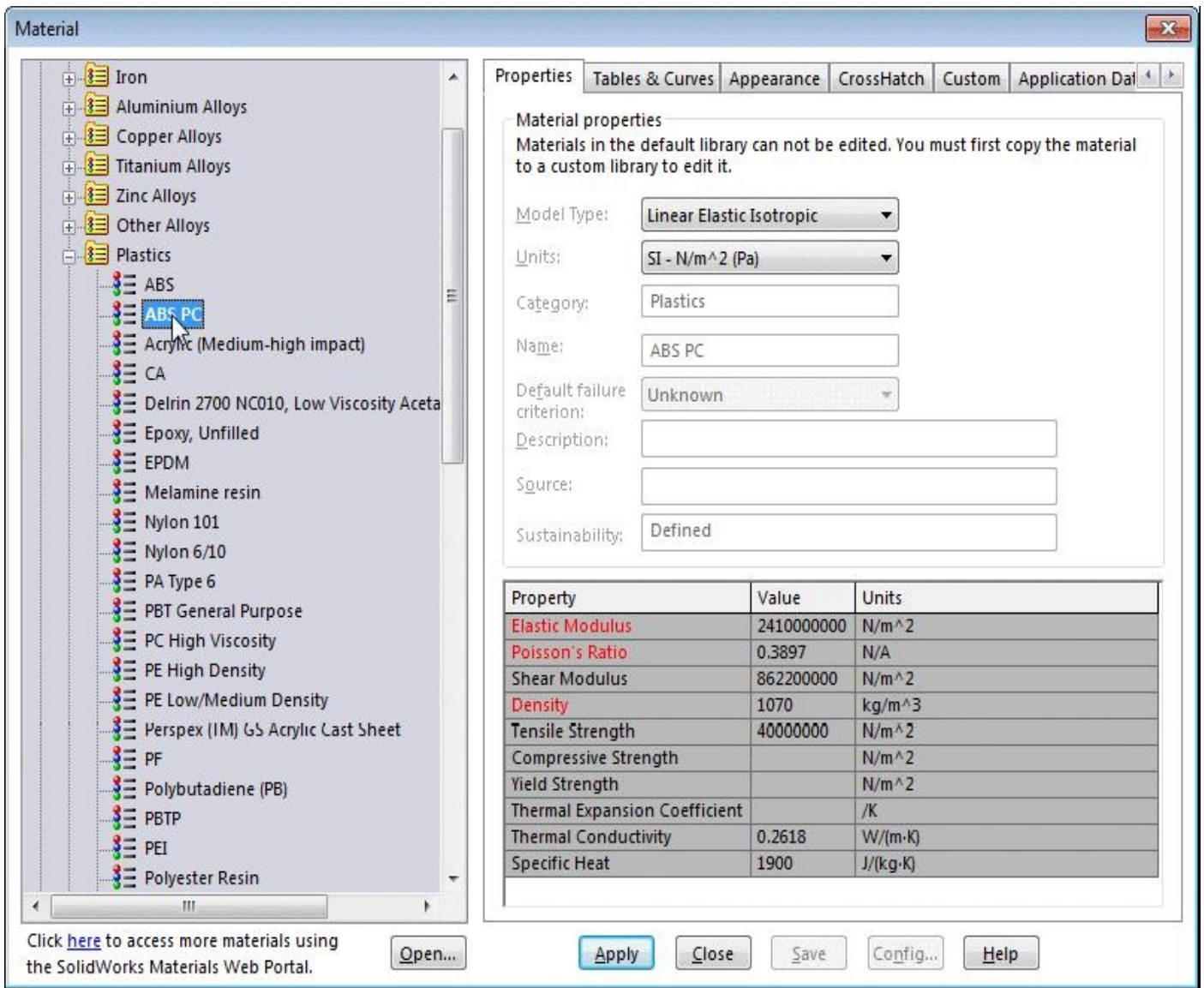


Figure-2. Material dialog box with material to be selected

- Click on the **Apply** button and then click on the **OK** button from the dialog box.

If you are working on an assembly then you can apply the contacts as discussed in earlier chapters.

Creating Mesh

- Right-click on the **Mesh** in the **Analysis Manager**. A shortcut menu will be displayed.
- Click on the **Simplify Model for Meshing** button from the shortcut menu. The **Simplify** task pane will be displayed; refer to [Figure-3](#).

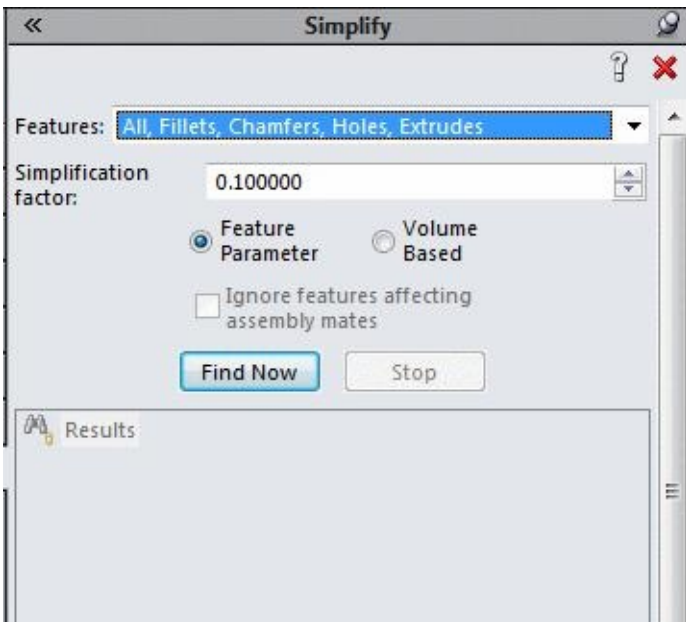


Figure-3. Simplify taskpane

- Click on the **Find Now** button to find the areas that create complexity in meshing. The complex areas are displayed below **Results** in the **Simplify** task pane; refer to [Figure-4](#).

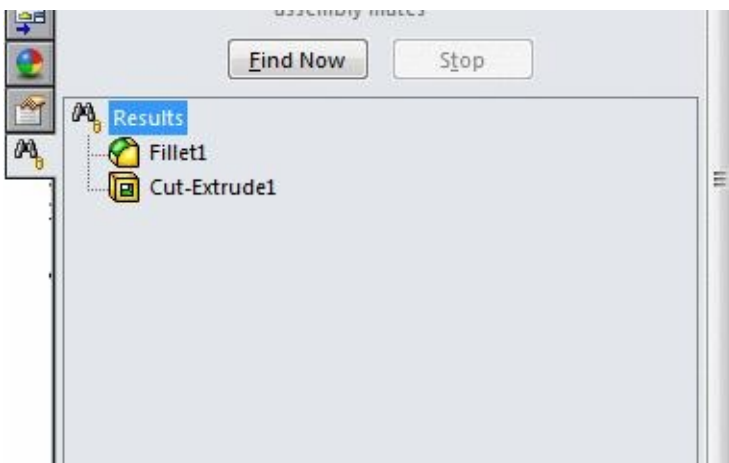


Figure-4. Results in task pane

- Close the **Simplify** task pane and select all the elements by holding **CTRL** key while selecting from the **FeatureManager**.
- Right-click on any of the selected element and select the **Suppress** button; refer to [Figure-5](#).

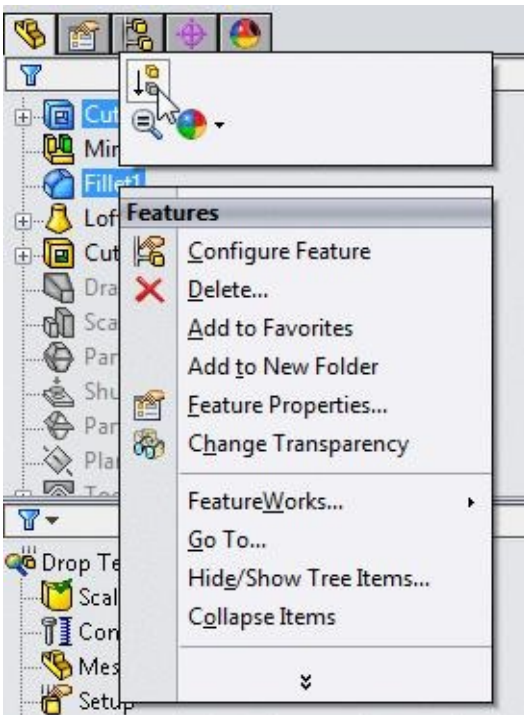


Figure-5. Suppress button

- On doing so, the selected elements will be suppressed.
- Again, right-click on **Mesh** in the **Analysis Manager** and select the **Create Mesh** button. The **Mesh PropertyManager** will be displayed; refer to [Figure-6](#).

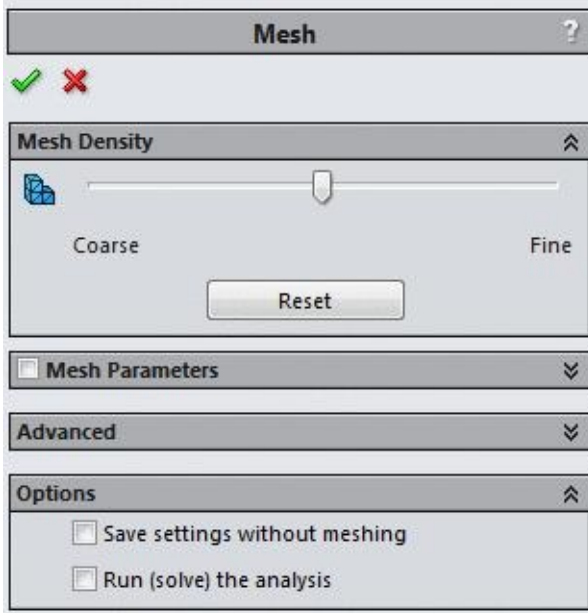


Figure-6. Mesh PropertyManager

- Set the mesh density as per your system capabilities and accuracy requirement.
- Click on the **OK** button from the **PropertyManager** to create the mesh.

Parameters for Drop

- Right-click on the **Setup** node from the **Analysis Manager**. A shortcut menu will be displayed; refer to [Figure-7](#).

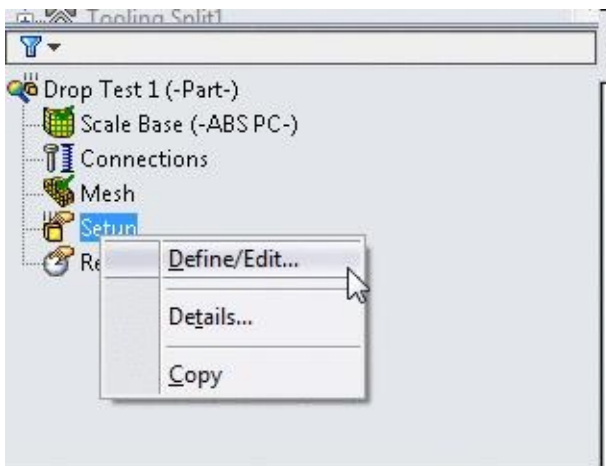


Figure-7. Setup shortcut menu

- Click on the **Define/Edit** button from the menu. The **Drop Test Setup PropertyManager** will be displayed; refer to [Figure-8](#).

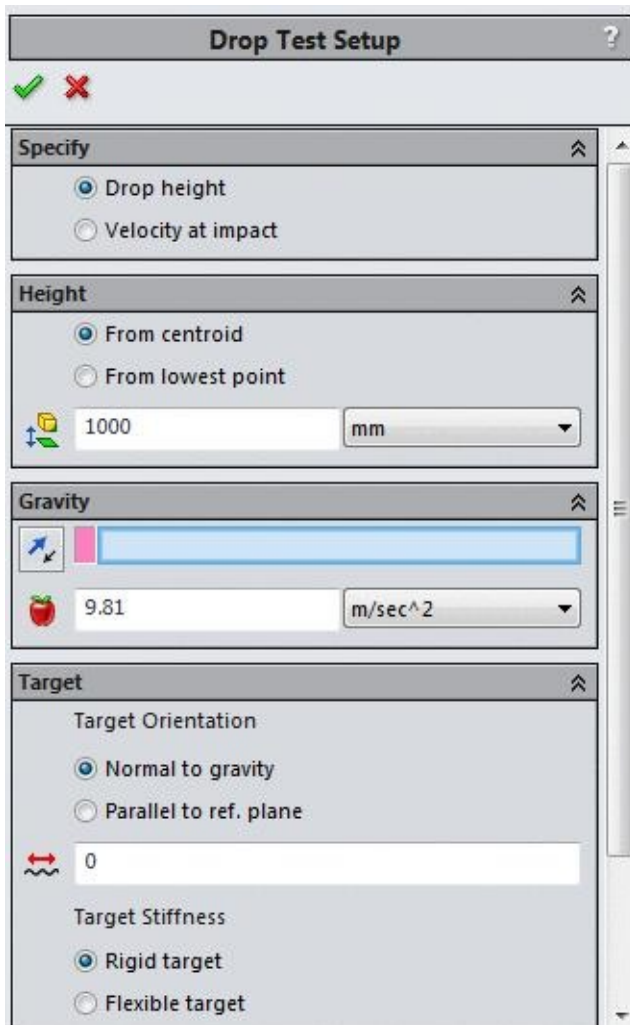


Figure-8. Drop Test Setup PropertyManager

- If you know the velocity of the object being dropped then click on the **Velocity at**

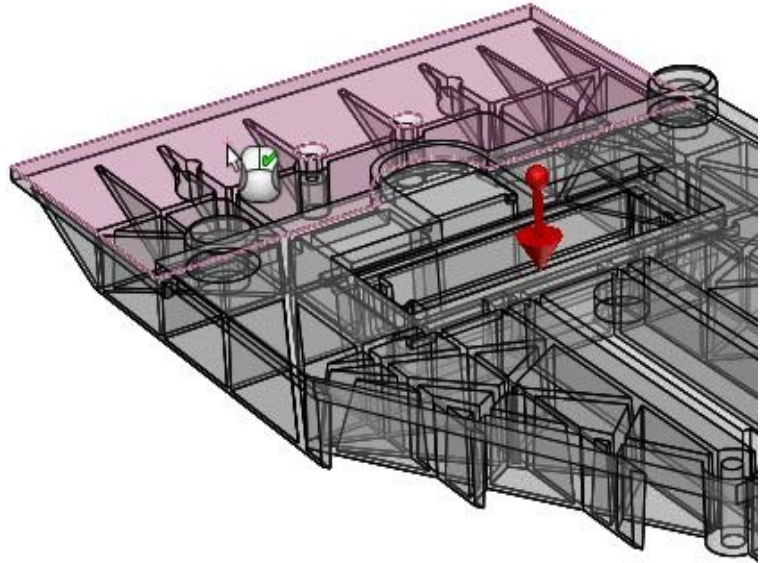
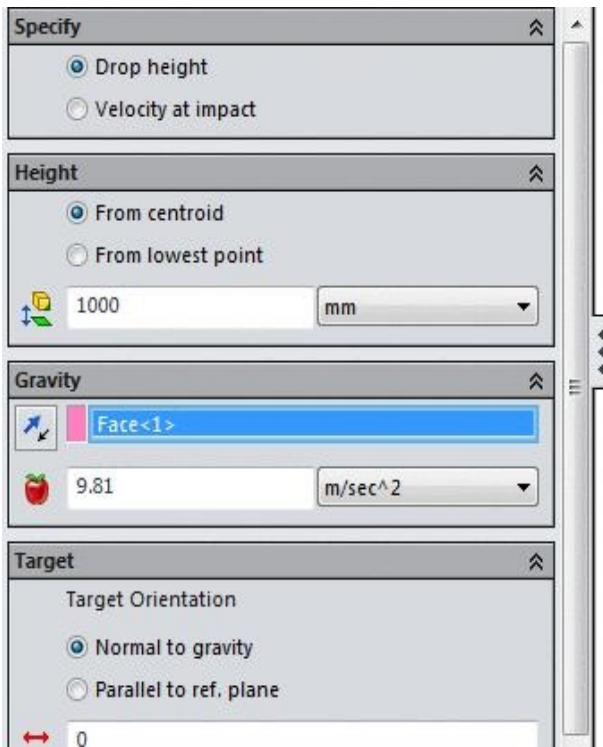


Figure-11. Face selected for gravity reference

- You can change the gravitational acceleration as per your requirement by using the **Gravity Magnitude** edit box in the **Gravity** rollout, if required.
- Select the **Normal to gravity** radio button in the **Target** rollout if the target plane (ground) is normal to gravity direction.
- Select the **Parallel to ref. plane** radio button and select the reference plane for specifying the ground orientation as per your requirement.
- You can specify the friction coefficient for selected material by using the **Friction Coefficient** edit box, if required.
- If the floor is tough then select the **Rigid target** radio button. If your floor is soft then select the **Flexible target** radio button and specify the related parameters in the **Stiffness and Thickness** rollout displayed in the **PropertyManager**.
- The edit box in **Contact Damping** rollout is used to specify the damping ratio for collision.
- After specifying the desired parameters, click on the **OK** button from the **PropertyManager** to apply the settings.

Running the Analysis

- Click on the **Run** button from the **Ribbon**. System will start solving the analysis.
- After the analysis is solved, the results will be displayed; refer to [Figure-12](#).

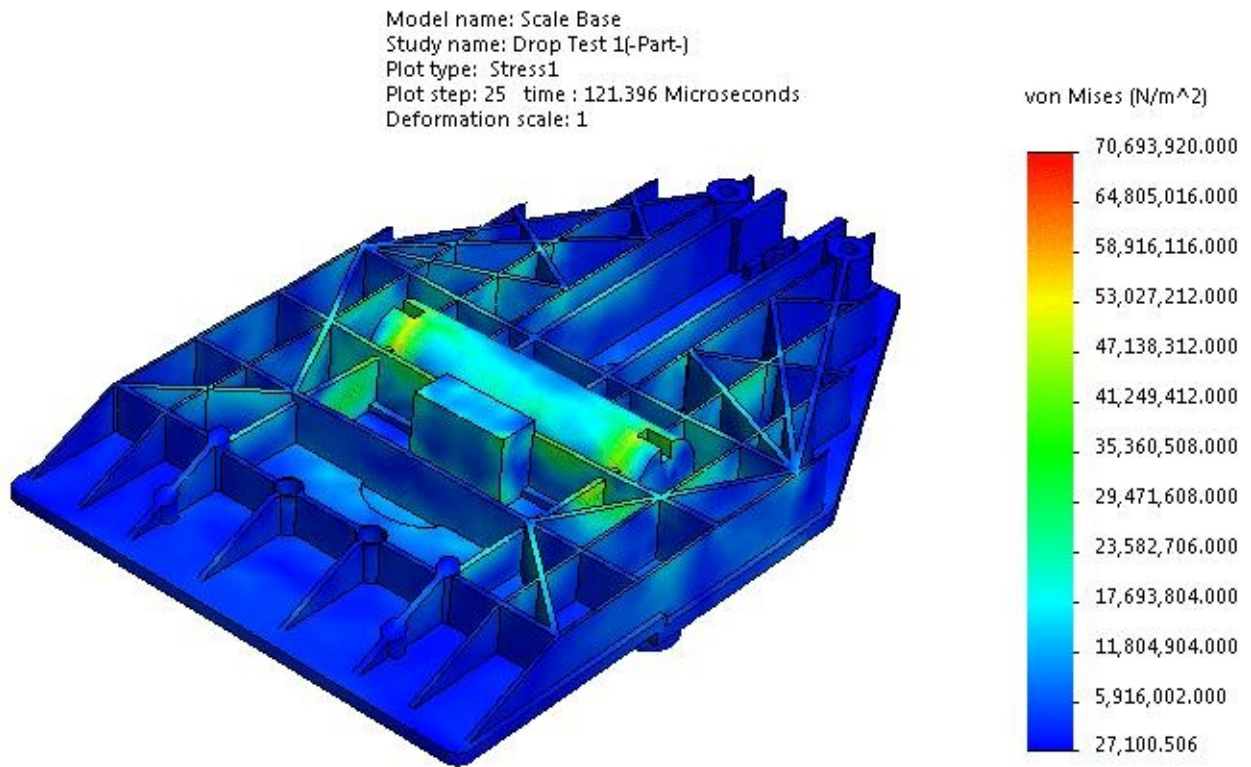


Figure-12. Results

- You can create more plots by right-clicking on **Results** node in the **Analysis Manager** and selecting the desired result option.

PRESSURE VESSEL DESIGN AND DESIGN STUDY

CHAPTER 12

Topics Covered

The major topics covered in this chapter are:

- *Starting Pressure Vessel Design.*
- *Inclusion of Analyses*
- *Simulating Design Study*
- *Starting Design Study*
- *Specifying Variables*
- *Setting Constraints*
- *Setting Goals*
- *Evaluating results*

INTRODUCTION

Pressure vessels are containers that store the pressurized fluids. A pressure vessel can be

of any practical shape but generally spherical, cylindrical, and conical shapes are employed. The most important question for pressure vessel designer is whether the designed vessel will sustain the required conditions or not. SolidWorks Simulation provides a separate option to perform this study named **Pressure Vessel Design** study.

In a **Pressure Vessel Design** study, you combine the results of static studies with the desired factors. Each static study has a different set of loads that produce corresponding results. These loads can be dead loads, live loads (approximated by static loads), thermal loads, seismic loads, and so on. The Pressure Vessel Design study combines the results of the static studies algebraically using a linear combination or the square root of the sum of the squares (SRSS).

Before we start performing the Pressure Vessel Design study, there must be two or more static analyses performed on the vessel earlier. In the next example, we have already performed two static analyses and one thermal analysis on the model. Details of these analyses are given as follows:

Thermal Analysis 1

Parameters for thermal analysis are given in [Figure-1](#).

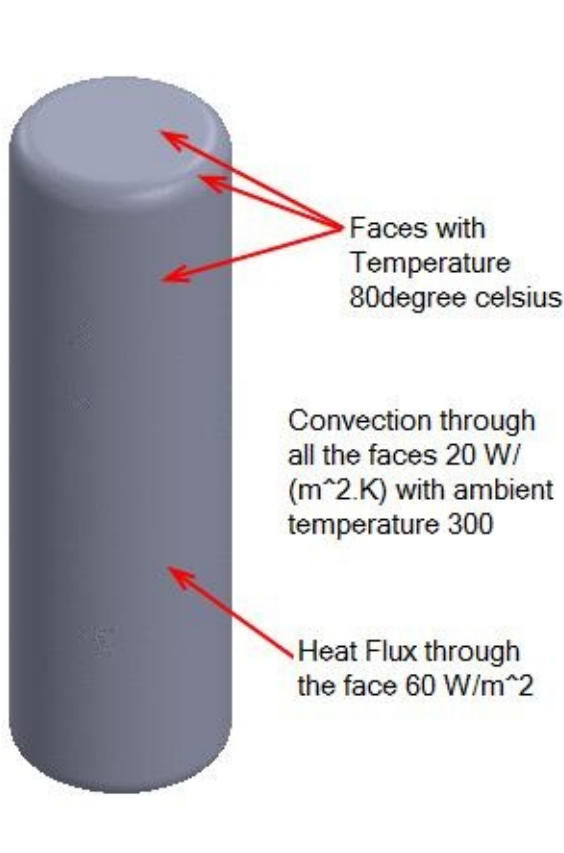


Figure-1. Parameters for thermal analysis

You can perform the analysis as per the above figure for your practice. Note that the

analysis is already performed in the part file supplied in the Resource kit.

Static Analysis 1 & 2

In the [Figure-2](#), the parameters for static analysis 1 are given and in the [Figure-3](#) parameters for static analysis 2 are given.

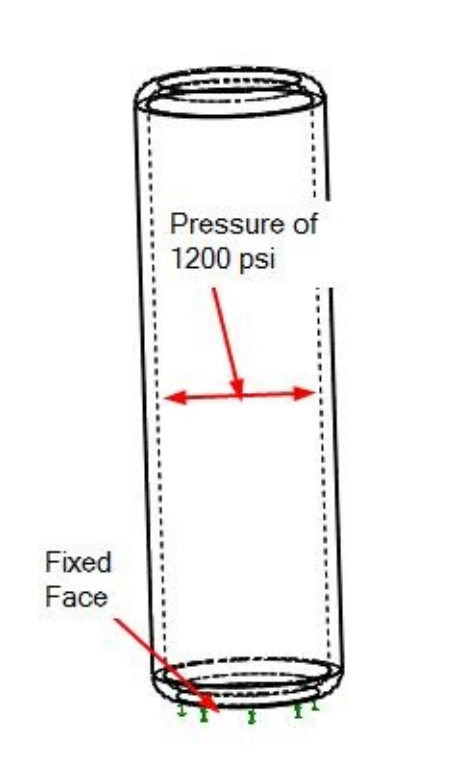


Figure-2. Parameters for static analysis 1

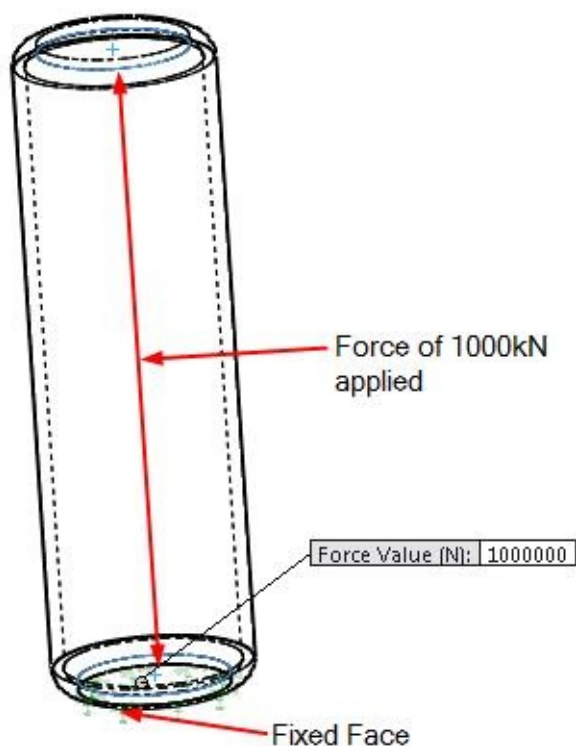


Figure-3. Parameters for static analysis 2

STARTING PRESSURE VESSEL DESIGN STUDY

- Open the file on which you have performed the above given analyses.
- Click on the **Simulation** tab in the **Ribbon** and click on the down arrow below **Study Advisor**. List of tools will be displayed.
- Click on the **New Study** button and select the **Pressure Vessel Design** button from the **PropertyManager**.
- Click on the **OK** button from the **PropertyManager** to display the tools related to **Pressure Vessel Design Study**; refer to [Figure-4](#).

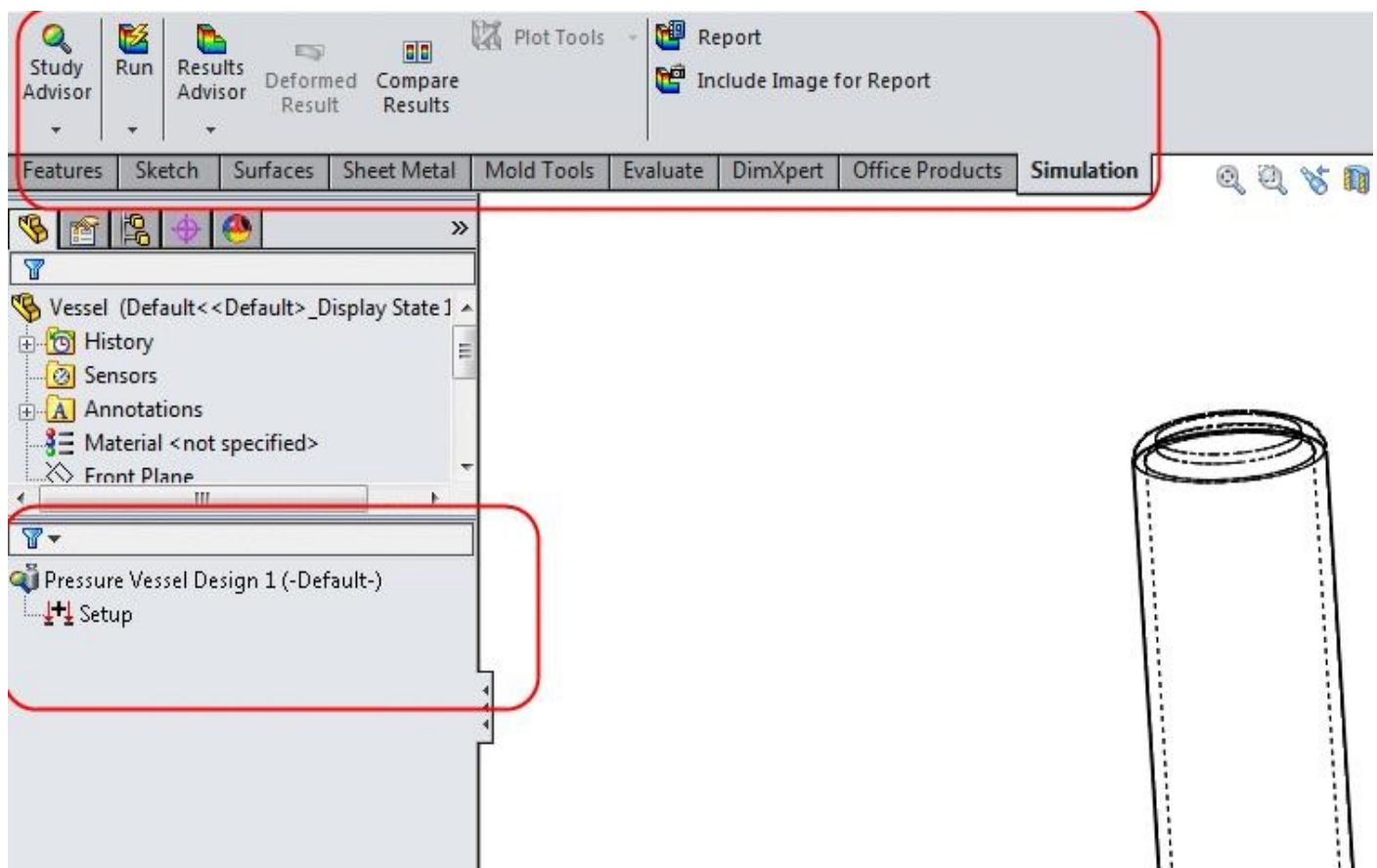


Figure-4. Tools for pressure vessel design

Setup for Design Study

- Right click on the **Setup** node in the **Analysis Manager**. A shortcut menu will be displayed; refer to [Figure-5](#).



Figure-5. Shortcut menu for setup

- Click on the **Define/edit** button. The **Result Combination Setup PropertyManager** will be displayed; refer to [Figure-6](#).

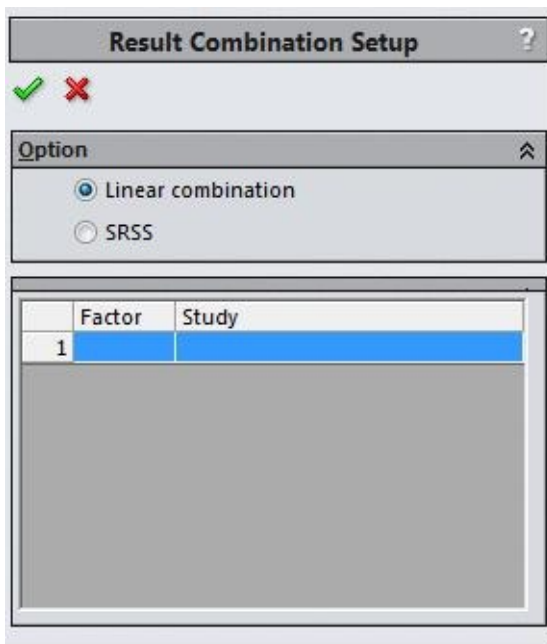


Figure-6. Result Combination Setup PropertyManager

There are two ways to combine various analyses results to study pressure vessel design; Linear combination and SRSS. Select the **Linear combination** radio button to linearly add the results of various studies to display final result. If you select the **SRSS** radio button then the square root of the sum of the squares study results will be used to represent the final result.

- Click in the field below **Study** column, the field will be converted to drop-down.
- Click on the down arrow of drop-down. The list of static analyses studies will displayed; refer to [Figure-7](#).

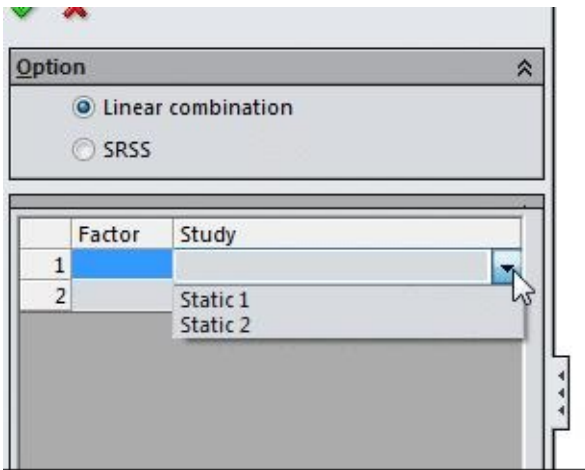


Figure-7. List of studies

- Select the **Static 1** study from the list. **1** factor will automatically appear in the **Factor** column adjacent to the study and one more row will be added below the selected field.
- Click on the empty field under **Study** column and select the **Static 2** study from the drop-down as discussed earlier.
- Click on the **OK** button from the **PropertyManager**.

Running the Design Study

- Click on the **Run** button in the **Ribbon**. The combining process of two studies will start.
- After the study is complete, the results will be displayed in the **Result** node of the **Analysis Manager**; refer to [Figure-8](#).

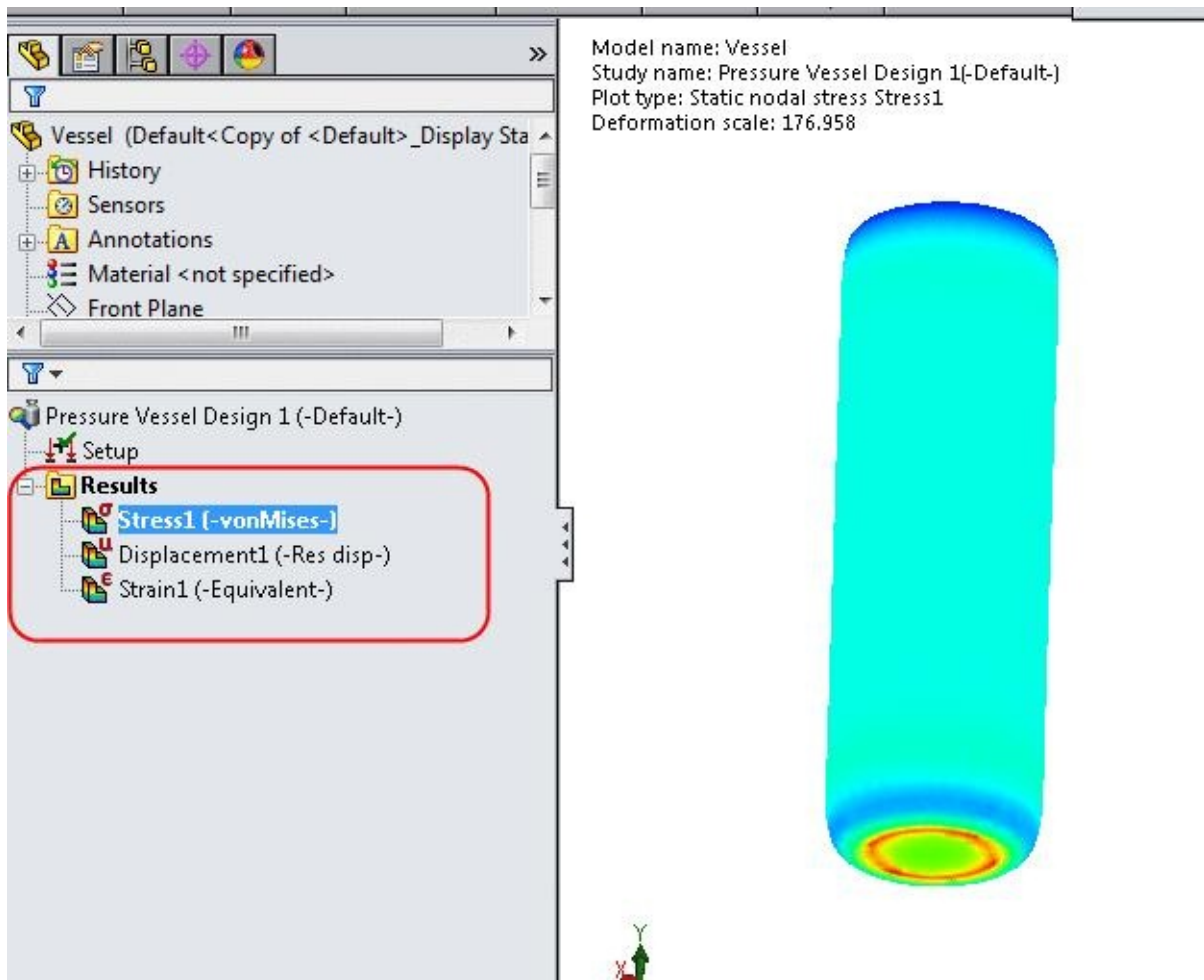


Figure-8. Results after combination

- Right-click on the **Results** node and select the **Define Factor of Safety Plot** button. The **Factor of Safety PropertyManager** will be displayed; refer to [Figure-9](#).



Figure-9. Factor of safety PropertyManager

- Click on the **OK** button from the **PropertyManager**. The factor of safety plot will be created in the modeling area; refer to [Figure-10](#).

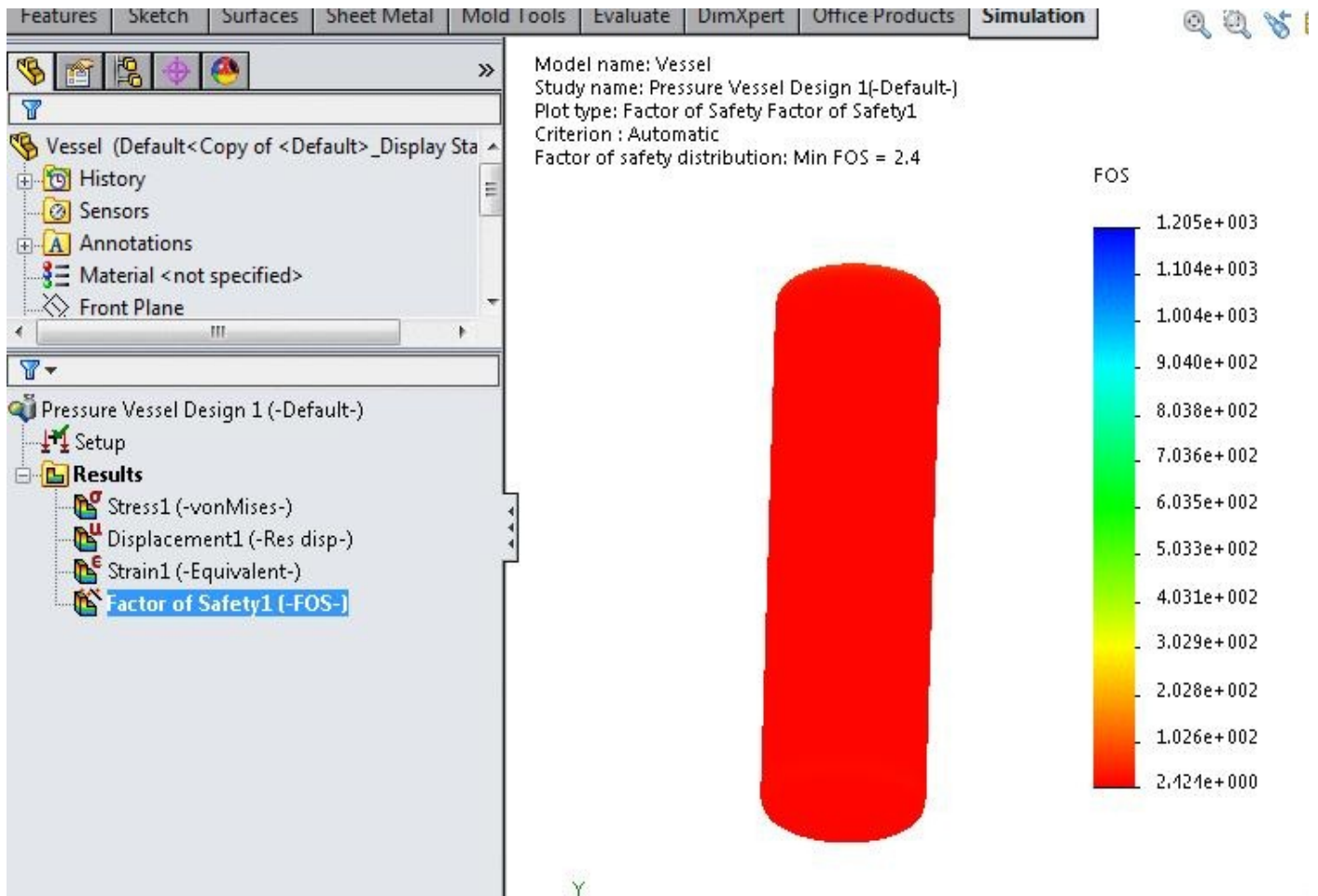


Figure-10. Factor of safety plot

DESIGN STUDY

Design Study is used to perform the optimization of the model. There are various parameters of an object that decide whether the model under testing will sustain or fail under the given condition. Out of all the parameters of a model, there are a few parameters of our concern that we want to use for satisfying our objective. In a Design Study, our objective can be:

- Reducing the mass of the model.
- Decreasing the total manufacturing cost of the model.
- Increasing the strength of the model and so on.

To achieve these objective, we need to manipulate with dimensions, materials, process and so on. In SolidWorks Simulation Design Study is used to perform an optimization or evaluate specific scenarios of your design.

You can plot the updated bodies and the calculated results for different iterations or scenarios by selecting their columns on the Results View tab.

You tackle a vast number of problems using a design study.

You can:

- Define multiple variables using any simulation parameter, or driving global variable.
- Define multiple constraints using sensors.
- Define multiple goals using sensors.
- Analyze models without simulation results. For example, you can minimize the mass of an assembly with the variables, density and model dimensions, and the constraint, volume.
- Evaluate design choices by defining a parameter that sets bodies to use different materials as a variable.

The procedure to use the **Design Study** is given next.

Before starting the design study, it is important to understand that the design study is a kind of macro which performs a specific type of analysis with a varying values of a specific parameters. For example, we have performed a static analysis on a tube under a specific pressure and we find that its factor of safety is 64. Then, we will perform a design study to find out the thickness of tube which is optimum i.e. factor of safety is greater than 1 and thickness of tube is minimum possible. In the next example, we have performed such a test on the model.

Starting Design Study

- Open the model on which you have performed the relevant analysis. In this example, we have performed a static analysis and we have found the results as given in [Figure-11](#).

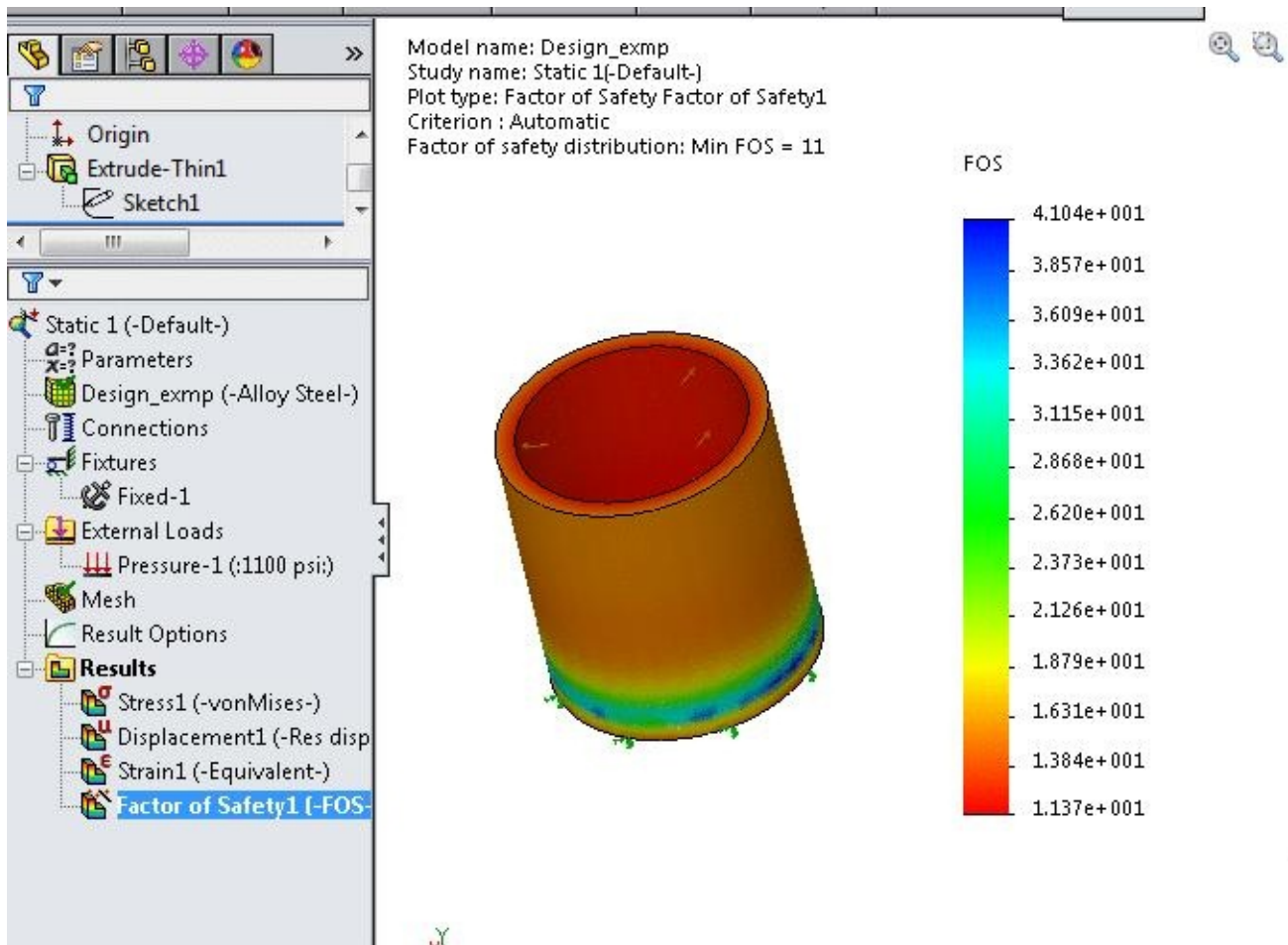


Figure-11. Factor of safety of example

- From the above figure, you can find that the factor of safety of the model under the given test is **11**.
- If we check the dimensions then the thickness of the model is 7 mm; refer to [Figure-12](#).

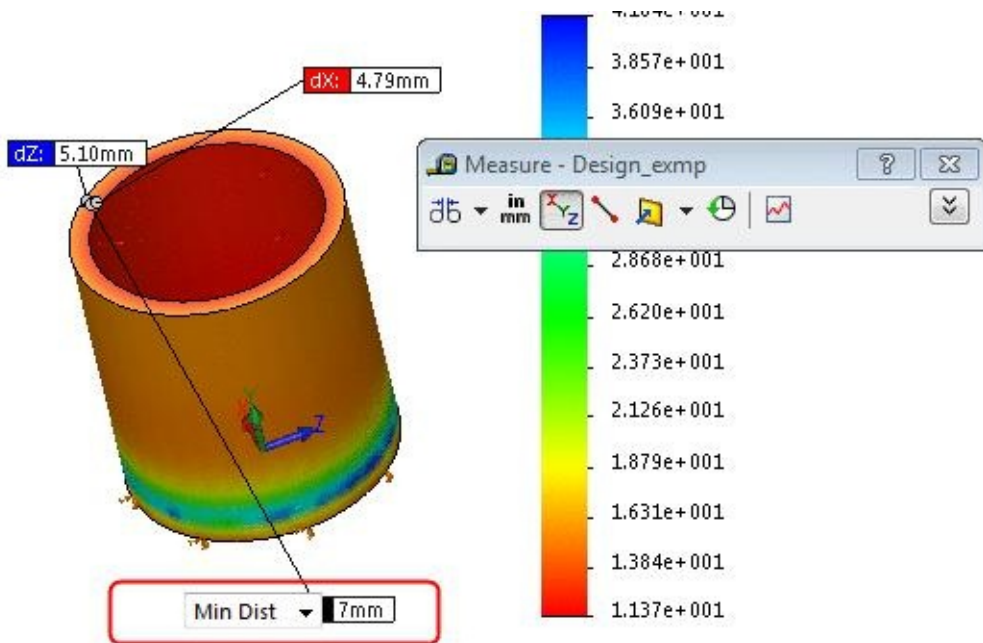


Figure-12. Thickness of tube

- Now, click on the **Simulation** tab in the **Ribbon**. The tools related to analysis will display.
- Click on the down arrow below **Study Advisor** and select the **New Study** tool from the list of tools displayed. The **Study PropertyManager** will be displayed.
- Click on the **Design Study** button and click on the **OK** button from the **PropertyManager**.
- The **Design Study** pane will be displayed; refer to [Figure-13](#).

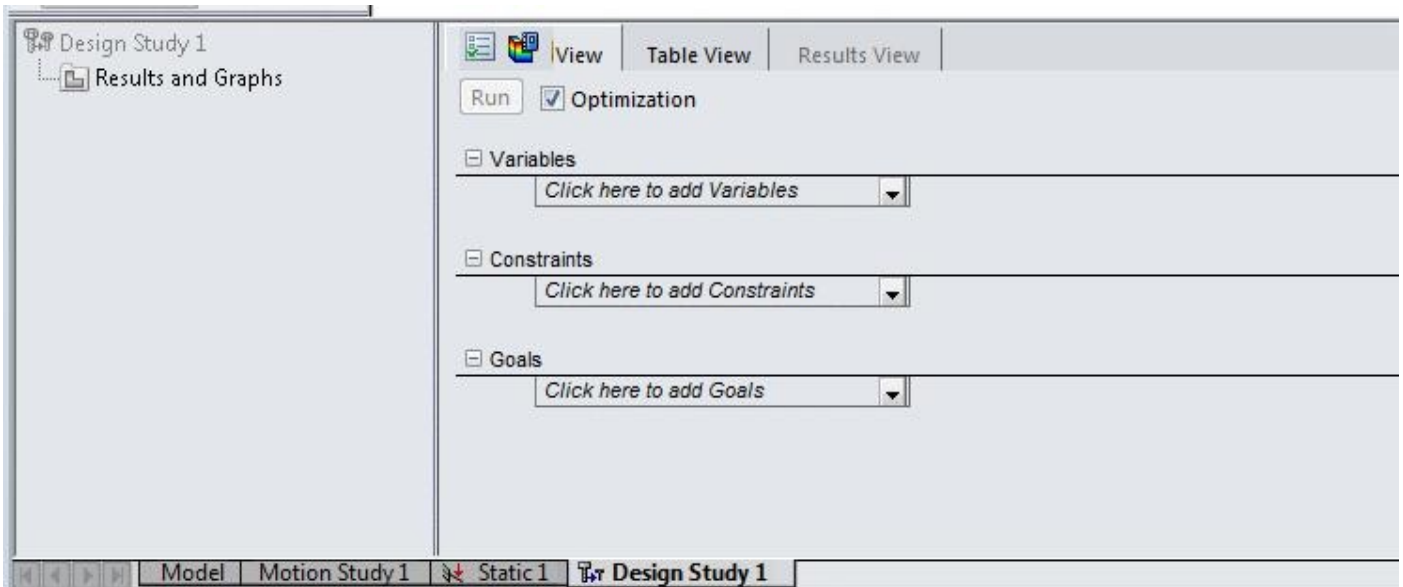


Figure-13. Design study pane

Setting options for Design Study

- Click on the **Design Study Options** button from the **Design Study** pane; refer to [Figure-14](#). The **Design Study Properties PropertyManager** will be displayed; refer to [Figure-15](#).

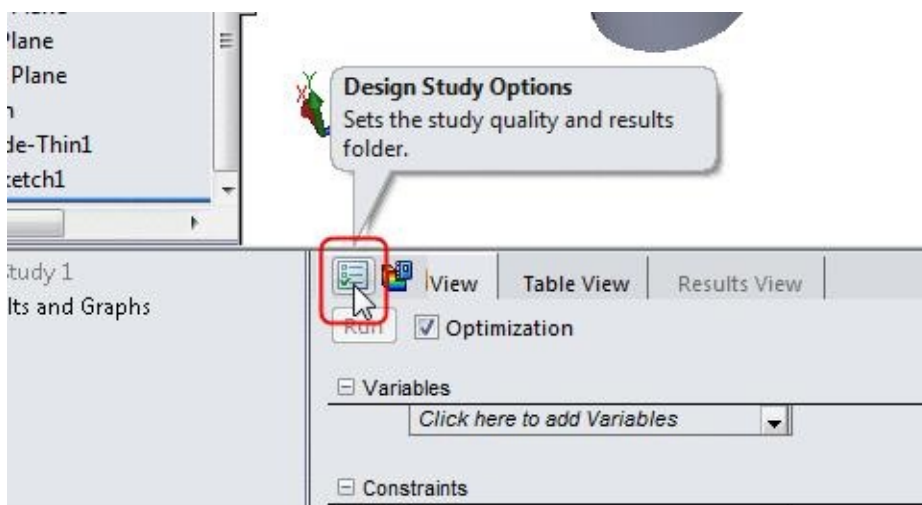


Figure-14. Design Study Options button

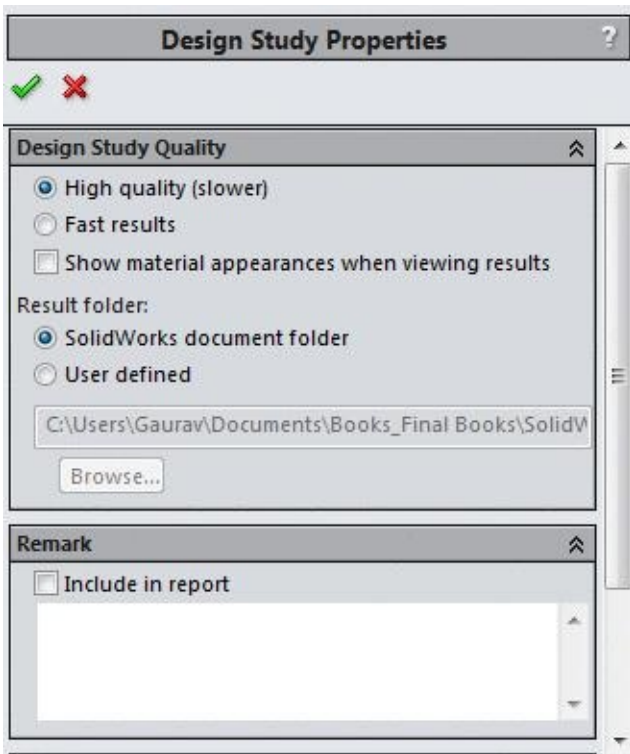


Figure-15. Design Study Properties PropertyManager

- Select the required radio button and set the desired parameters like; for fast results, click on the **Fast results** radio button.

Specifying parameters that are to be changed

- Click on the down arrow under the **Variables** node. A list of variable options will be displayed; refer to [Figure-16](#).

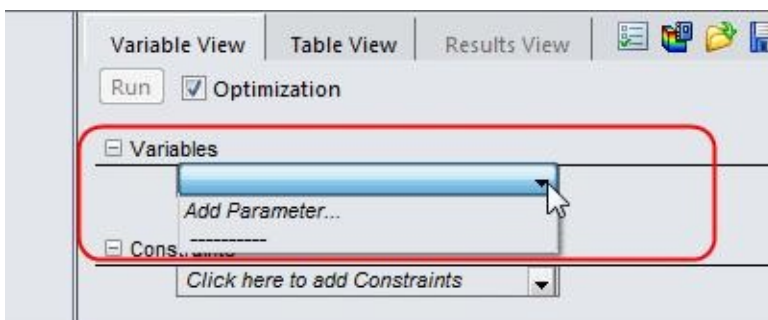


Figure-16. Variables drop-down

- Click on the **Add Parameter** option from the drop-down list. The **Parameters** dialog box will be displayed and you will be prompted to select a dimension that is to be varied; refer to [Figure-17](#).

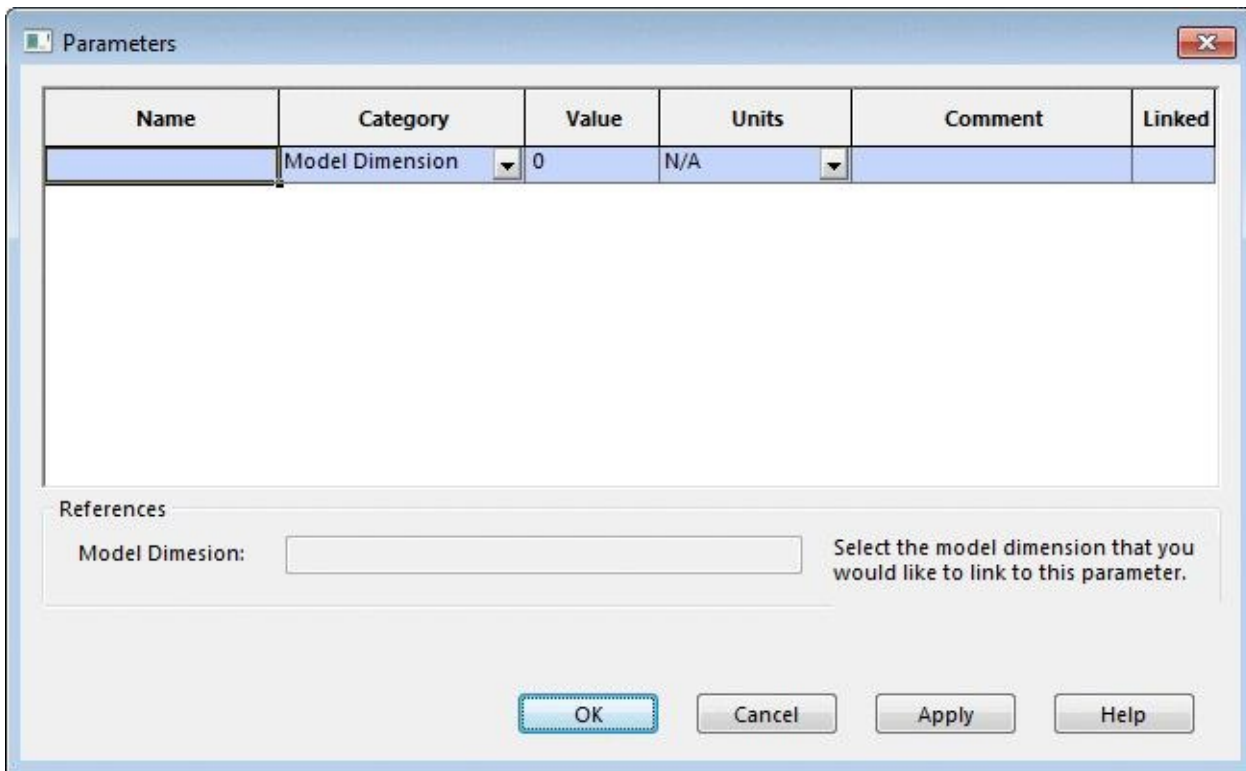


Figure-17. Parameters dialog box

- Click on the dimension of the model that you want to make variable; refer to [Figure-18](#). The selected dimension will be added under the **Value** column in the **Parameters** dialog box.

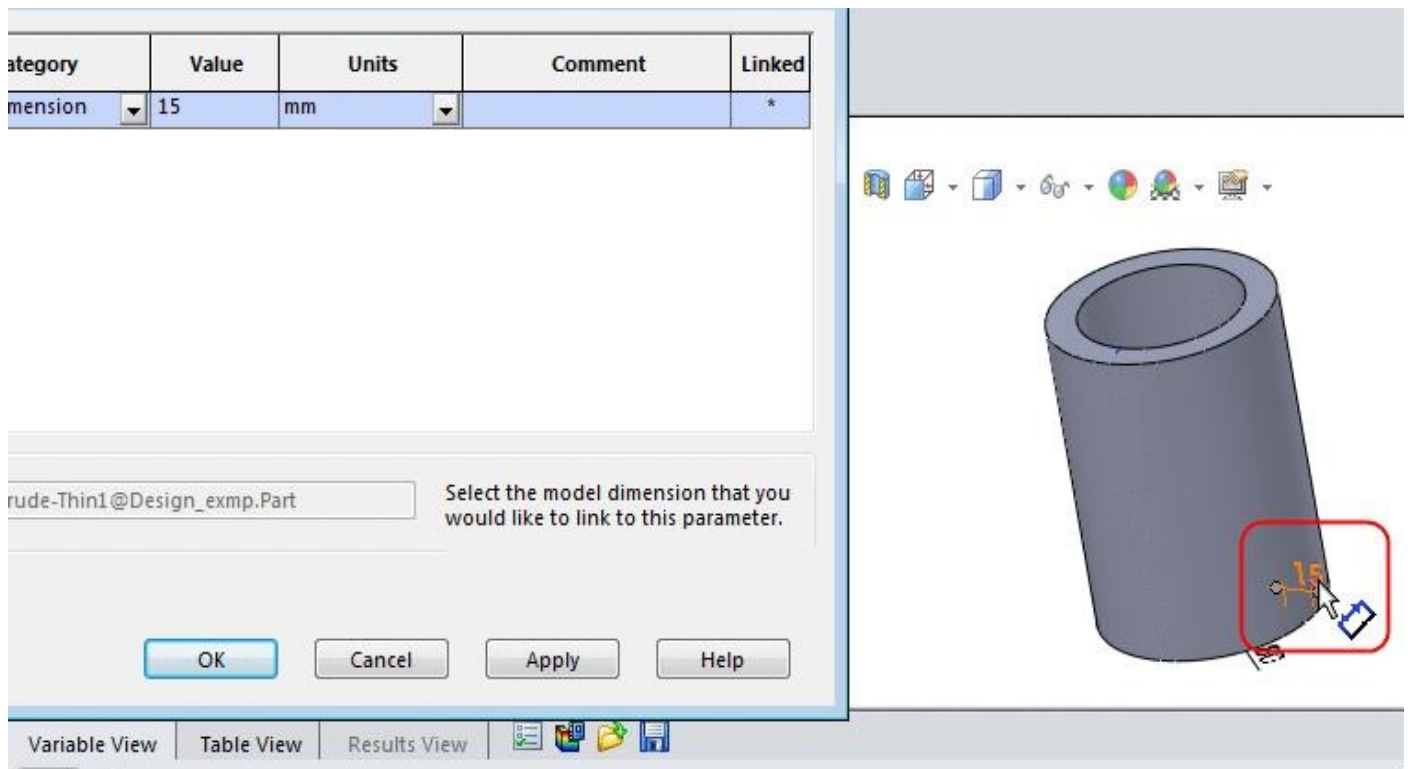


Figure-18. Dimension selected as variable

- Click in the field below **Name** column in the dialog box and specify the name as

Thickness; refer to [Figure-19](#).

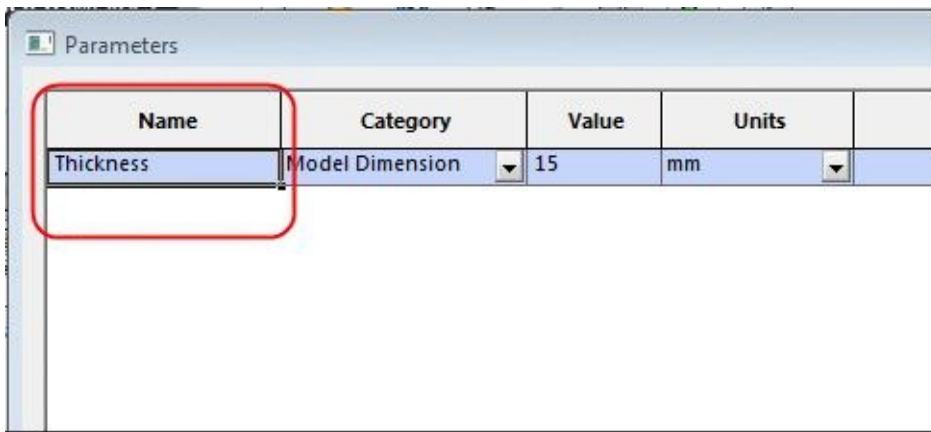


Figure-19. Name of parameter

- If you want to specify, then click in the name field of second row and follow the same procedure. You can also specify other parameters in spite of dimensions. To do so, click on the down arrow next to **Model Dimension** in **Category** column and select the desired category. There are four categories from which you can specify the variables: **Model Dimension**, **Global Variable**, **Simulation** data, and **Material** data.
- After specifying the variables, click on the **OK** button from the dialog box. The variable will be added in the **Design Study** pane and its range will be displayed; refer to [Figure-20](#).

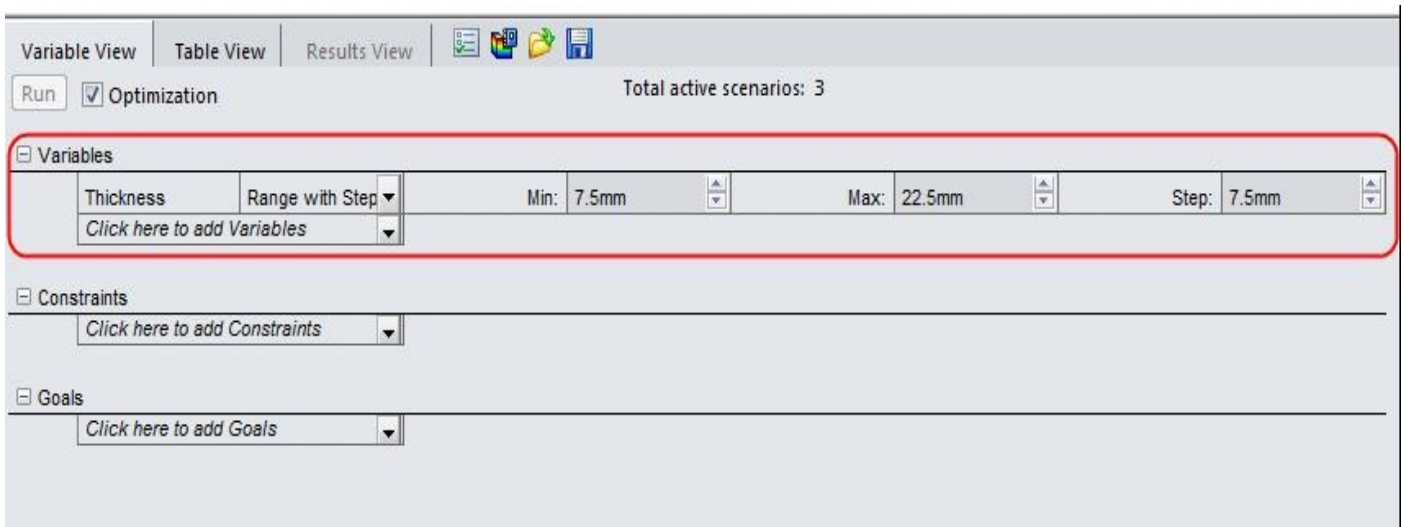


Figure-20. Design variable with range

- We know that the thickness of our model is **7 mm** with **Factor of Safety** as **11**. Now, we need to move below this thickness value to get the optimum thickness of the tube.
- Change the value of **Min**, **Max**, and **Step** as shown in [Figure-21](#) by using the spinners.

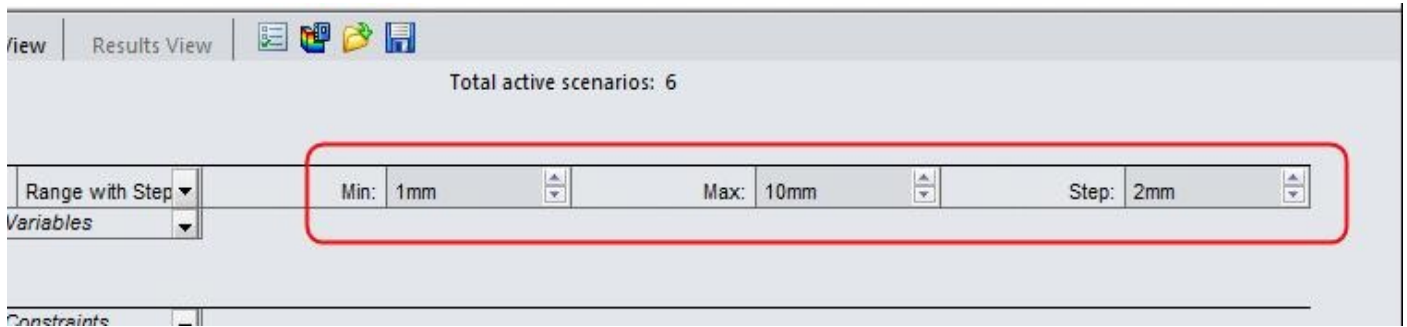


Figure-21. Values to be changed

Specifying restrictions for changes

- Click on the down arrow under the **Constraints** node in the **Design Study** pane. A list of options will be displayed.
- Click on the **Add Sensor** option from the list. The **Sensor PropertyManager** will be displayed; refer to [Figure-22](#).

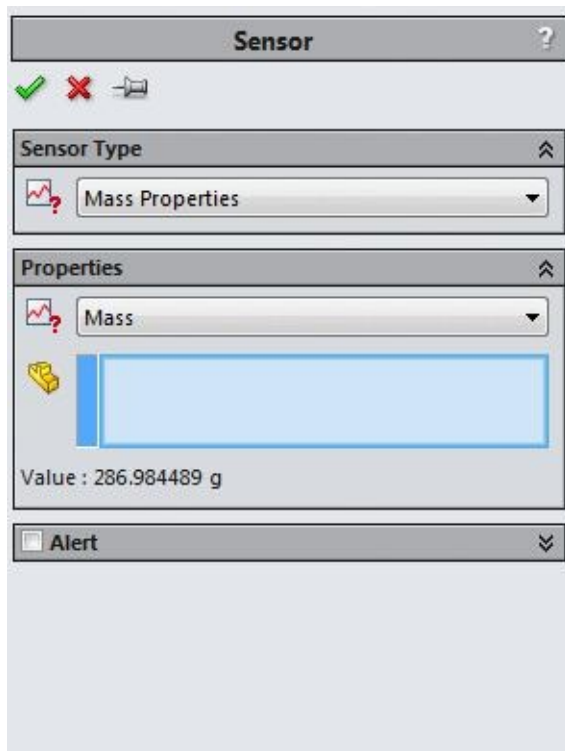


Figure-22. Sensor PropertyManager

- Click on the drop-down in the **Sensor Type** rollout. A list of options will be displayed.
- Select the **Simulation Data** option from the list; refer to [Figure-23](#).

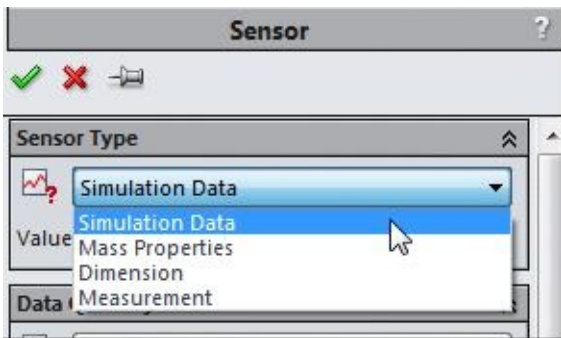


Figure-23. Simulation Data option

- On doing so, the options related to simulation data will be displayed in the **PropertyManager**.
- Click on the **Results** drop-down in the **Data Quantity** rollout and select the **Factor of Safety** option; refer to [Figure-24](#).

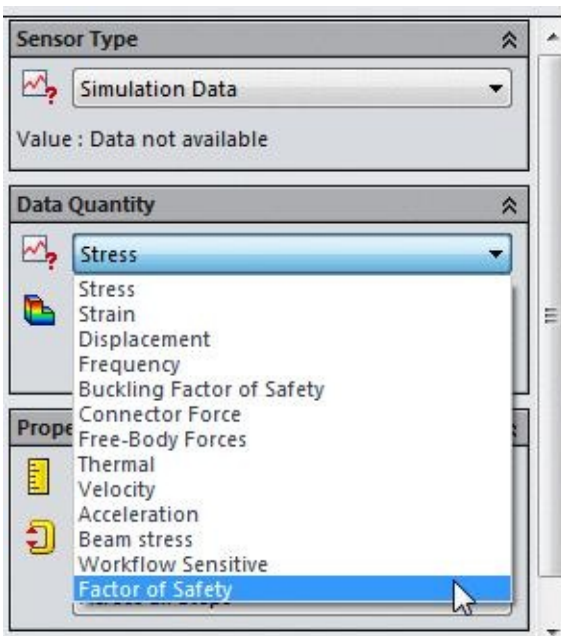


Figure-24. Factor of safety option

- Click on the **OK** button from the **Sensor PropertyManager** to add the **Factor of Safety** constraint. The constraint will be added in the **Design Study** pane; refer to [Figure-25](#).

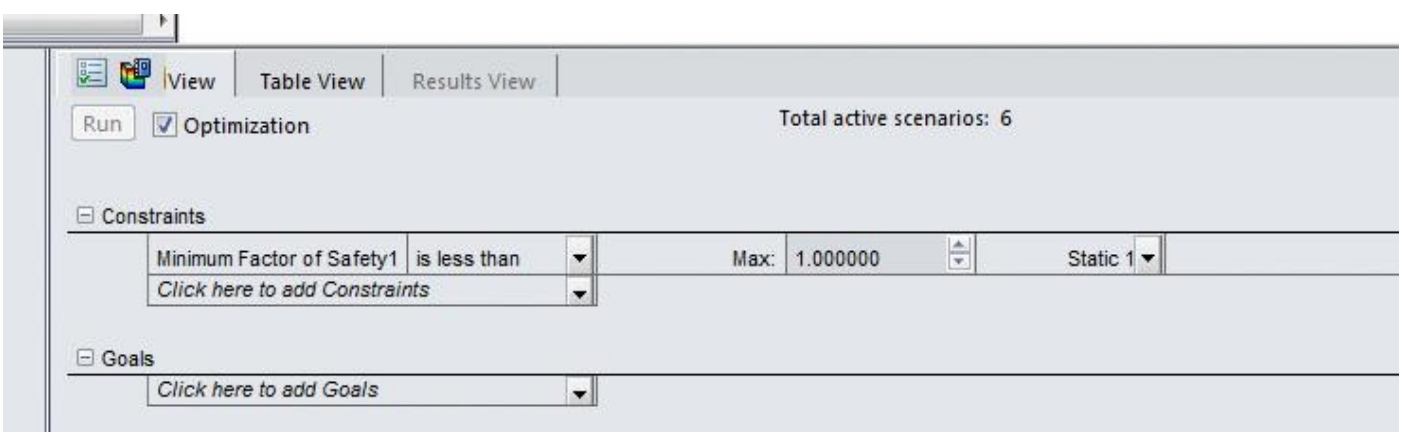


Figure-25. Factor of safety in Design Study pane

- Click on the down arrow next to **is less than** field and select the **Is greater than** option; refer to [Figure-26](#).

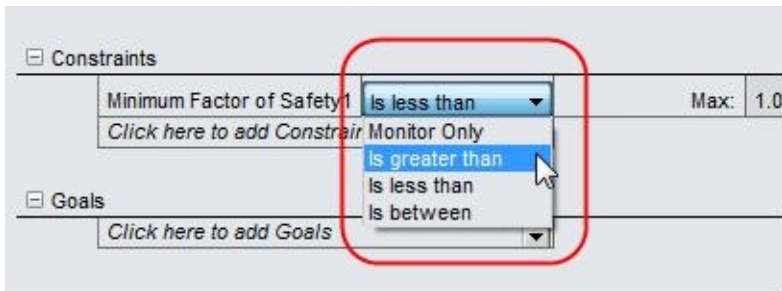


Figure-26. Option to be selected

Setting Goal for the study

- Click on the down arrow under the **Goals** node. A list of options will be displayed; refer to [Figure-27](#).

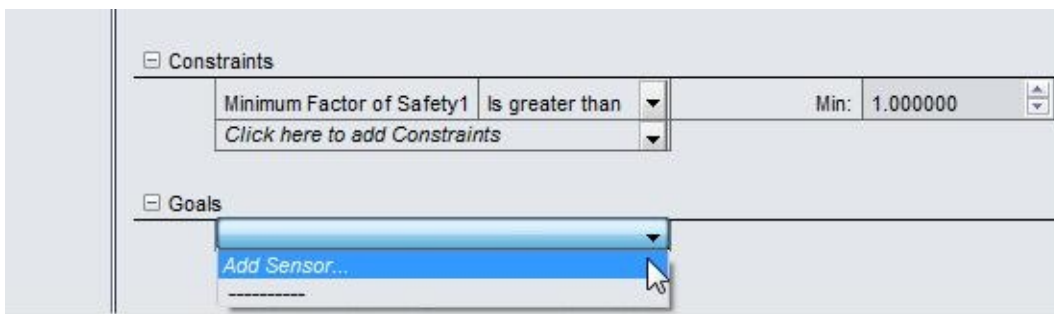


Figure-27. Goals drop down

- Click on the **Add Sensor** option. The **Sensor PropertyManager** will be displayed as discussed earlier; refer to [Figure-28](#).

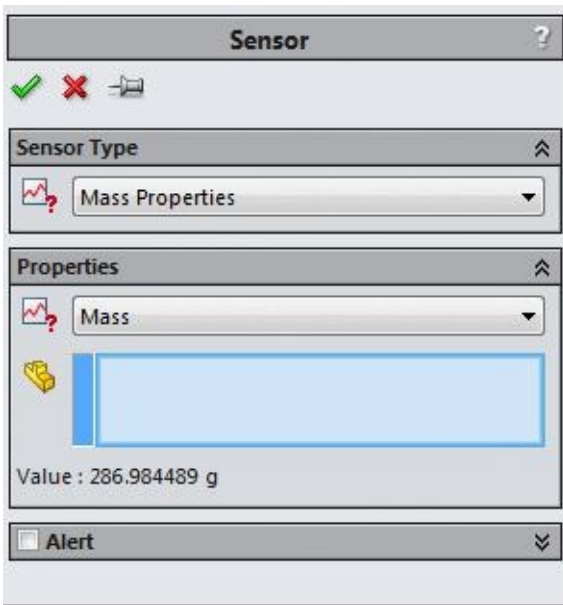


Figure-28. Sensor PropertyManager for goals

- Click on the **OK** button from the **PropertyManager** to set the mass to be minimized. The **Mass** will be added as goal in the **Design Study** pane; refer to [Figure-29](#).

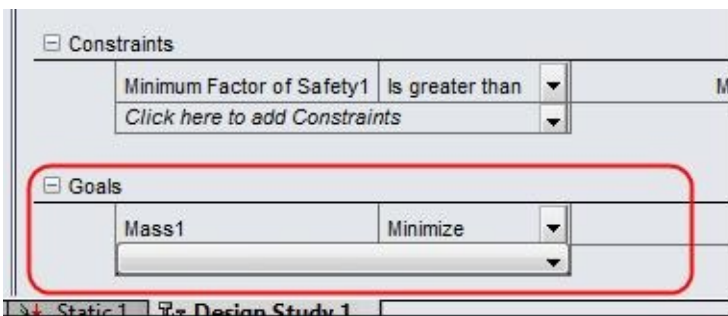


Figure-29. Goal added in Design Study

Running the Study

We have specified all the necessary parameters for the study and are ready for the analysis.

- Click on the **Run** button in the **Design Study** pane, the design study will start.
- After the study is solved, the results will be displayed as shown in [Figure-30](#).

		Current	Initial	Optimal (1)	Scenario 1	Scenario 2	Scenario 3
Thickness	<input type="text" value="10mm"/>	10mm	10mm	1mm	1mm	5mm	8mm
Minimum Factor of Safety1	> 1.000000	56.079132	56.079132	5.597877	5.597877	30.756149	46.716949
Mass1	Minimize	424.115 g	424.115 g	38.1704 g	38.1704 g	200.277 g	331.752 g

Figure-30. Results of design study

- The green highlighted box displays the optimum result for the analysis and its related parameters used for optimization.

You can specify more variables for changing the design and test the results for practice.

FUNDAMENTALS OF FEA

CHAPTER 13



Topics Covered

The major topics covered in this chapter are:

- ***Introduction to FEA.***
- ***General Description of FEA***
- ***Solving a problem with FEA***
- ***FEA V/s Classical Methods***
- ***FEA V/s Finite Different Method***
- ***Need for Studying FEM***
- ***Refining Mesh***
- ***Higher order element V/s Refined Mesh***
- ***Optimum process of FEA through SolidWorks Simulation***

INTRODUCTION

The finite element analysis is a numerical technique. In this method all the complexities of the problems, like varying shape, boundary conditions and loads are maintained as they are but the solutions obtained are approximate. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering. The fast improvements in computer hardware technology and slashing of cost of computers have boosted this method, since the computer is the basic need for the application of this method. A number of popular brand of finite element analysis packages are now available commercially. Some of the popular packages are SolidWorks Simulation, NASTRAN, NISA, and ANSYS. Using these packages one can analyse several complex structures.

The finite element analysis originated as a method of stress analysis in the design of aircrafts. It started as an extension of matrix method of structural analysis. Today this method is used not only for the analysis in solid mechanics, but even in the analysis of fluid flow, heat transfer, electric and magnetic fields and many others. Civil engineers use this method extensively for the analysis of beams, space frames, plates, shells, folded plates, foundations, rock mechanics problems and seepage analysis of fluid through porous media. Both static and dynamic problems can be handled by finite element analysis. This method is used extensively for the analysis and design of ships, aircraft, space crafts, electric motors and heat engines.

GENERAL DESCRIPTION OF THE METHOD

In engineering problems there are some basic unknowns. If they are found, the behavior of the entire structure can be predicted. The basic unknowns or the Field variables which are encountered in the engineering problems are displacements in solid mechanics, velocities in fluid mechanics, electric and magnetic potentials in electrical engineering and temperatures in heat flow problems.

In a continuum, these unknowns are infinite. The finite element procedure reduces such unknowns to a finite number by dividing the solution region into small parts called elements and by expressing the unknown field variables in terms of assumed approximating functions (Interpolating functions/Shape functions) within each element. The approximating functions are defined in terms of field variables of specified points called nodes or nodal points. Thus in the finite element analysis, the unknowns are the field variables of the nodal points. Once these are found the field variables at any point can be found by using interpolation functions.

After selecting elements and nodal unknowns next step in finite element analysis is to assemble element properties for each element. For example, in solid mechanics, we have to find the force-displacement i.e. stiffness characteristics of each individual element.

Mathematically this relationship is of the form

$$[k]_e \{\delta\}_e = \{F\}_e$$

where $[k]_e$ is element stiffness matrix, $\{\delta\}_e$ is nodal displacement vector of the element and $\{F\}_e$ is nodal force vector. The element of stiffness matrix k_{ij} represent the force in coordinate direction 'i' due to a unit displacement in coordinate direction 'j'. Four methods are available for formulating these element properties viz. direct approach, variational approach, weighted residual approach and energy balance approach. Any one of these methods can be used for assembling element properties. In solid mechanics variational approach is commonly employed to assemble stiffness matrix and nodal force vector (consistent loads).

Element properties are used to assemble global properties/structure properties to get system equations $[k] \{\delta\} = \{F\}$. Then the boundary conditions are imposed. The solution of these simultaneous equations give the nodal unknowns. Using these nodal values additional calculations are made to get the required values e.g. stresses, strains, moments, etc. in solid mechanics problems.

Thus the various steps involved in the finite element analysis are:

- (i) Select suitable field variables and the elements.
- (ii) Discretise the continua.
- (iii) Select interpolation functions.
- (iv) Find the element properties.
- (v) Assemble element properties to get global properties.
- (vi) Impose the boundary conditions.
- (vii) Solve the system equations to get the nodal unknowns.
- (viii) Make the additional calculations to get the required values.

A BRIEF EXPLANATION OF FEA FOR A STRESS ANALYSIS PROBLEM

The steps involved in finite element analysis are clarified by taking the stress analysis of a

tension strip with fillets (refer Fig.1.1). In this problem stress concentration is to be studied in the fillet zone. Since the problem is having symmetry about both x and y axes, only one quarter of the tension strip may be considered as shown in Fig.1.2. About the symmetric axes, transverse displacements of all nodes are to be made zero. The various steps involved in the finite element analysis of this problem are discussed below:

Step 1: Four noded isoparametric element (refer Fig 1.3) is selected for the analysis (However note that 8 noded isoparametric element is ideal for this analysis). The four noded isoparametric element can take quadrilateral shape also as required for elements 12, 15, 18, etc. As there is no bending of strip, only displacement continuity is to be ensured but not the slope continuity. Hence displacements of nodes in x and y directions are taken as basic unknowns in the problem.

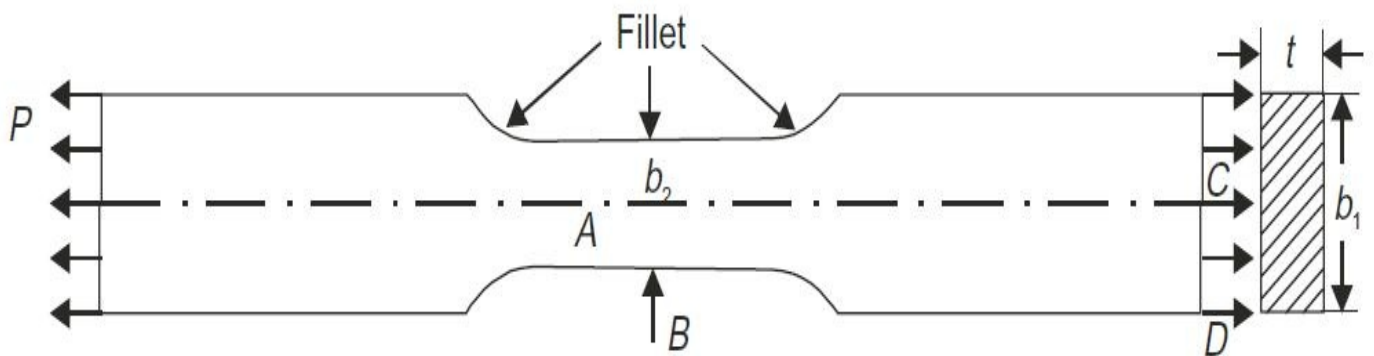


Fig. 1.1 Typical tension flat

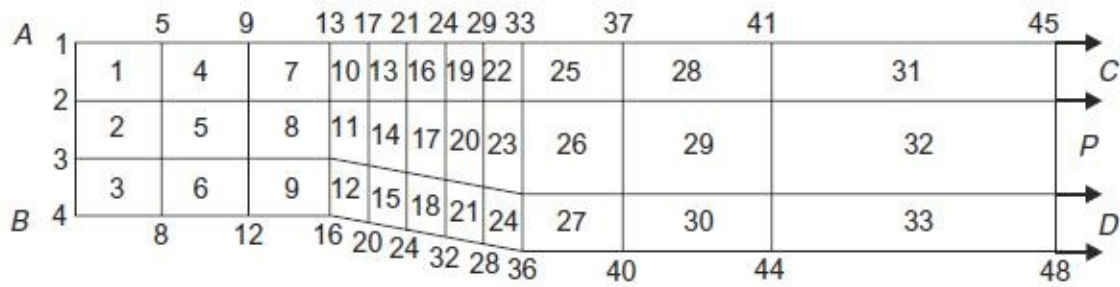


Fig. 1.2 Discretisation of quarter of tension flat

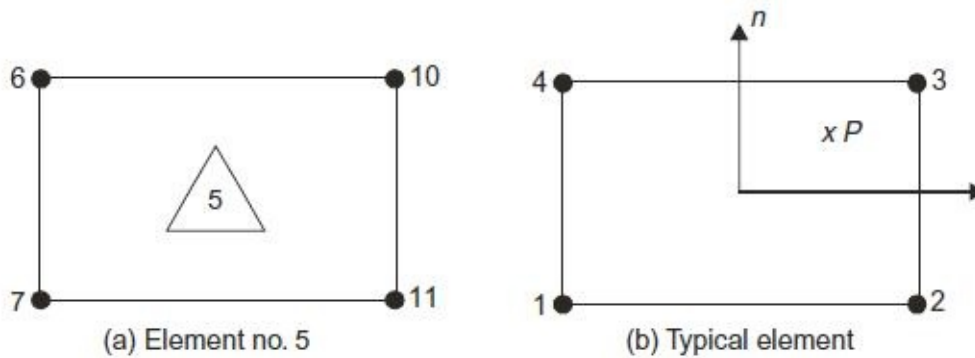


Fig. 1.3

Step 2: The portion to be analysed is to be discretised. Fig. 1.2 shows discretised portion. For this 33 elements have been used. There are 48 nodes. At each node unknowns are x and y components of displacements. Hence in this problem total unknowns (displacements) to be determined are $48 \times 2 = 96$.

Step 3: The displacement of any point inside the element is approximated by suitable functions in terms of the nodal displacements of the element. For the typical element (Fig. 1.3 b), displacements at P are

$$u = \sum N_i u_i = N_1 u_1 + N_2 u_2 + N_3 u_3 + N_4 u_4$$

$$\text{and } v = \sum N_i v_i = N_1 v_1 + N_2 v_2 + N_3 v_3 + N_4 v_4 \dots (1.2)$$

The approximating functions N_i are called shape functions or interpolation functions. Usually they are derived using polynomials.

Step 4: Now the stiffness characters and consistent loads are to be found for each element. There are four nodes and at each node degree of freedom is 2. Hence degree of freedom in each element is $4 \times 2 = 8$. The relationship between the nodal displacements and nodal forces is called element stiffness characteristics. It is of the form

$[k]_e \{\delta\}_e = \{F\}_e$, as explained earlier.

For the element under consideration, k_e is 8×8 matrix and δ_e and F_e are vectors of 8 values. In solid mechanics element stiffness matrix is assembled using variational approach i.e. by minimizing potential energy.

If the load is acting in the body of element or on the surface of element, its equivalent nodal points are to be found using variational approach, so that right hand side of the above expression is assembled. This process is called finding consistent loads.

Step 5: The structure is having $48 \times 2 = 96$ displacement and load vector components to be determined.

Hence global stiffness equation is of the form

$$[k] \{\delta\} = \{F\}$$

$$96 \times 96 \quad 96 \times 1 \quad 96 \times 1$$

Each element stiffness matrix is to be placed in the global stiffness matrix appropriately. This process is called assembling global stiffness matrix. In this problem force vector F is zero at all nodes except at nodes 45, 46, 47 and 48 in x direction. For the given loading nodal equivalent forces are found and the force vector F is assembled.

Step 6: In this problem, due to symmetry transverse displacements along AB and BC are zero. The system equation $[k] \{\delta\} = \{F\}$ is modified to see that the solution for $\{\delta\}$ comes out with the above values. This modification of system equation is called imposing the boundary conditions.

Step 7: The above 96 simultaneous equations are solved using the standard numerical procedures like Gauss elimination or Choleski's decomposition techniques to get the 96 nodal displacements.

Step 8: Now the interest of the analyst is to study the stresses at various points. In solid mechanics the relationship between the displacements and stresses are well established. The stresses at various points of interest may be found by using shape functions and the nodal displacements and then stresses calculated. The stress concentrations may be studied by comparing the values obtained at various points in the fillet zone with the values at uniform zone, far away from the fillet (which is equal to P/b_2t).

FINITE ELEMENT METHOD V/S CLASSICAL METHODS

1. In classical methods exact equations are formed and exact solutions are obtained where as in finite element analysis exact equations are formed but approximate solutions are obtained.

2. Solutions have been obtained for few standard cases by classical methods, where as solutions can be obtained for all problems by finite element analysis.

3. Whenever the following complexities are faced, classical method makes the drastic assumptions' and looks for the solutions:

(a) Shape

(b) Boundary conditions

(c) Loading

Fig. 1.4 shows such cases in the analysis of slabs (plates).

To get the solution in the above cases, rectangular shapes, same boundary condition along a side and regular equivalent loads are to be assumed. In FEM no such assumptions are made. The problem is treated as it is.

4. When material property is not isotropic, solutions for the problems become very difficult in classical method. Only few simple cases have been tried successfully by researchers. FEM can handle structures with anisotropic properties also without any difficulty.

5. If structure consists of more than one material, it is difficult to use classical method, but finite element can be used without any difficulty.

6. Problems with material and geometric non-linearities can not be handled by classical methods.

There is no difficulty in FEM.

Hence FEM is superior to the classical methods only for the problems involving a number

of complexities which cannot be handled by classical methods without making drastic assumptions. For all regular problems, the solutions by classical methods are the best solutions. Infact, to check the validity of the FEM programs developed, the FEM solutions are compared with the solutions by classical methods for standard problems.

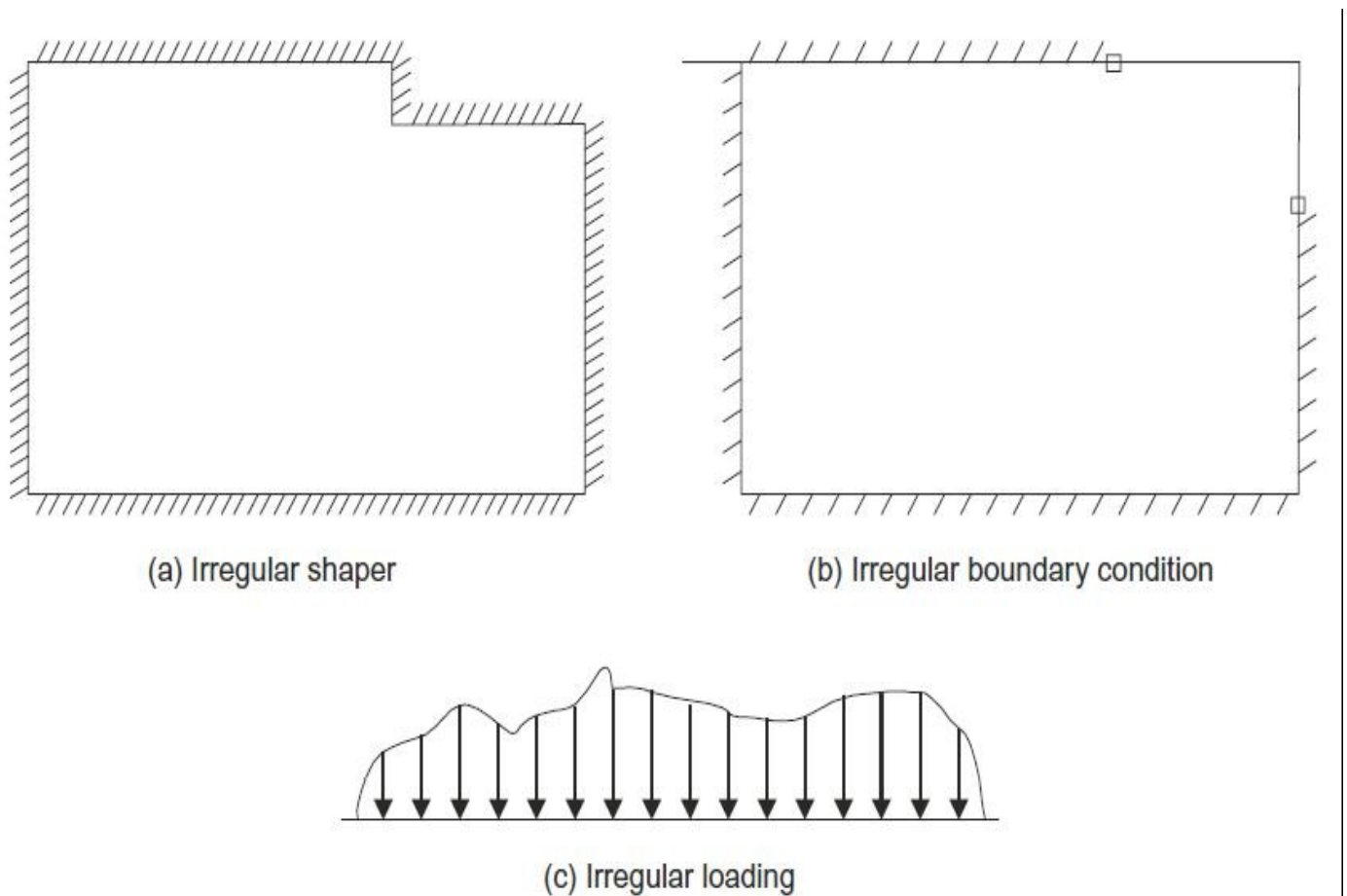


Fig. 1.4

FEM VS FINITE DIFFERENCE METHOD (FDM)

1. FDM makes pointwise approximation to the governing equations i.e. it ensures continuity only at the node points. Continuity along the sides of grid lines are not ensured. FEM make piecewise approximation i.e. it ensures the continuity at node points as well as along the sides of the element.

2. FDM do not give the values at any point except at node points. It do not give any approximating function to evaluate the basic values (deflections, in case of solid mechanics) using the nodal values. FEM can give the values at any point. However the values obtained at points other than nodes are by using suitable interpolation formulae.

3. FDM makes stair type approximation to sloping and curved boundaries as shown in Fig. 1.5. FEM can consider the sloping boundaries exactly. If curved elements are used, even the curved boundaries can be handled exactly.

4. FDM needs larger number of nodes to get good results while FEM needs fewer nodes.

5. With FDM fairly complicated problems can be handled where as FEM can handle all complicated problems.

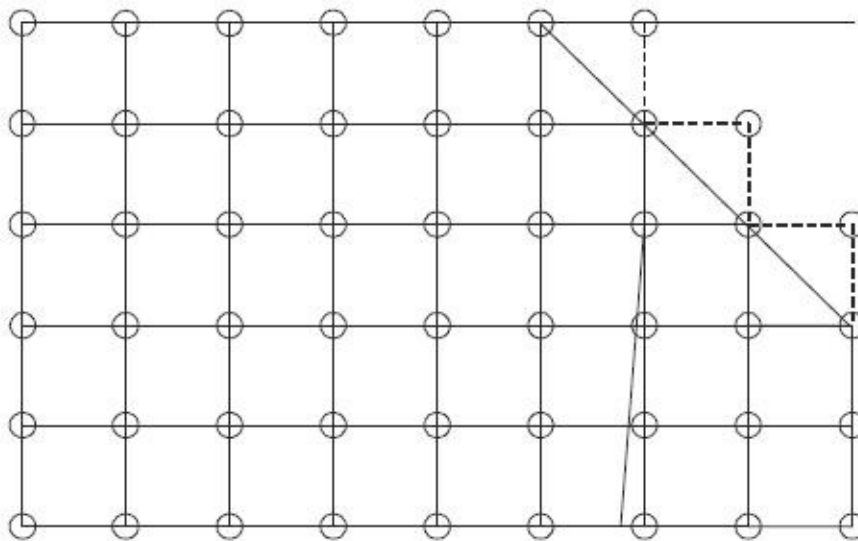


Fig. 1.5 FDM approximation of shape

NEED FOR STUDYING FEM

Now, a number of users friendly packages are available in the market. Hence one may ask the question ‘What is the need to study FEA?’.

The above argument is not sound. The finite element knowledge makes a good engineer better while just user without the knowledge of FEA may produce more dangerous results. To use the FEA packages properly, the user must know the following points clearly:

1. Which elements are to be used for solving the problem in hand.
2. How to discretise to get good results.
3. How to introduce boundary conditions properly.
4. How the element properties are developed and what are their limitations.
5. How the displays are developed in pre and post processor to understand their limitations.

6. To understand the difficulties involved in the development of FEA programs and hence the need for checking the commercially available packages with the results of standard cases.

Unless user has the background of FEA, he may produce worst results and may go with overconfidence. Hence it is necessary that the users of FEA package should have sound knowledge of FEA.

Warning To Fea Package Users

When hand calculations are made, the designer always gets the feel of the structure and get rough idea about the expected results. This aspect cannot be ignored by any designer, whatever be the reliability of the program, a complex problem may be simplified with drastic assumptions and FEA results obtained. Check whether expected trend of the result is obtained. Then avoid drastic assumptions and get more refined results with FEA package. User must remember that structural behaviour is not dictated by the computer programs. Hence the designer should develop feel of the structure and make use of the programs to get numerical results which are close to structural behaviour.

One of the main concern for using FEA applications is selection of elements, in other words discretization. The process of modeling a structure using suitable number, shape and size of the elements is called discretization. The modeling should be good enough to get the results as close to actual behavior of the structure as possible.

In a structure we come across the following types of discontinuities:

- (a) Geometric
- (b) Load
- (c) Boundary conditions
- (d) Material.

Geometric Discontinuities

Wherever there is sudden change in shape and size of the structure there should be a node or line of nodes. Figure 10.1 shows some of such situations.

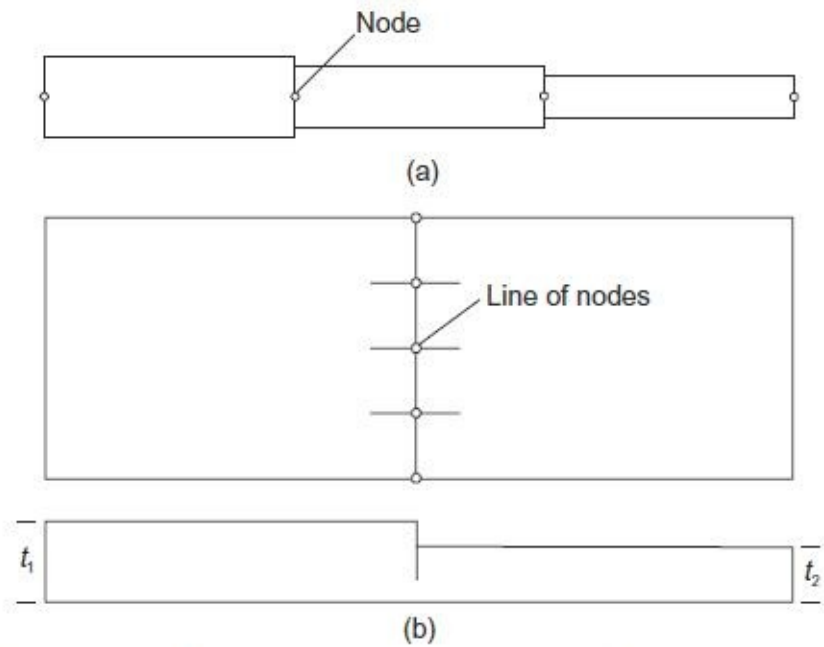


Fig. 10.1 (a) Bar subject to axial forces (b) Plate with varying

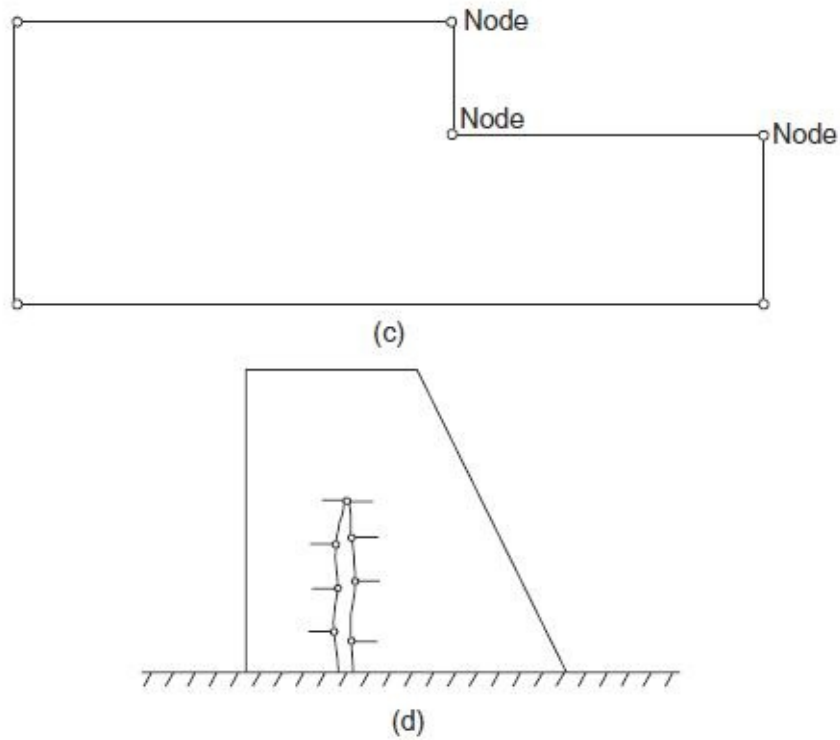


Fig. 10.1 (contd) (c) Slab (d) Dam with a crack

Discontinuity of Loads

Concentrated loads and sudden change in the intensity of uniformly distributed loads are the sources of discontinuity of loads. A node or a line of nodes should be there to model the structure. Some of these situations are shown in Fig. 10.2.

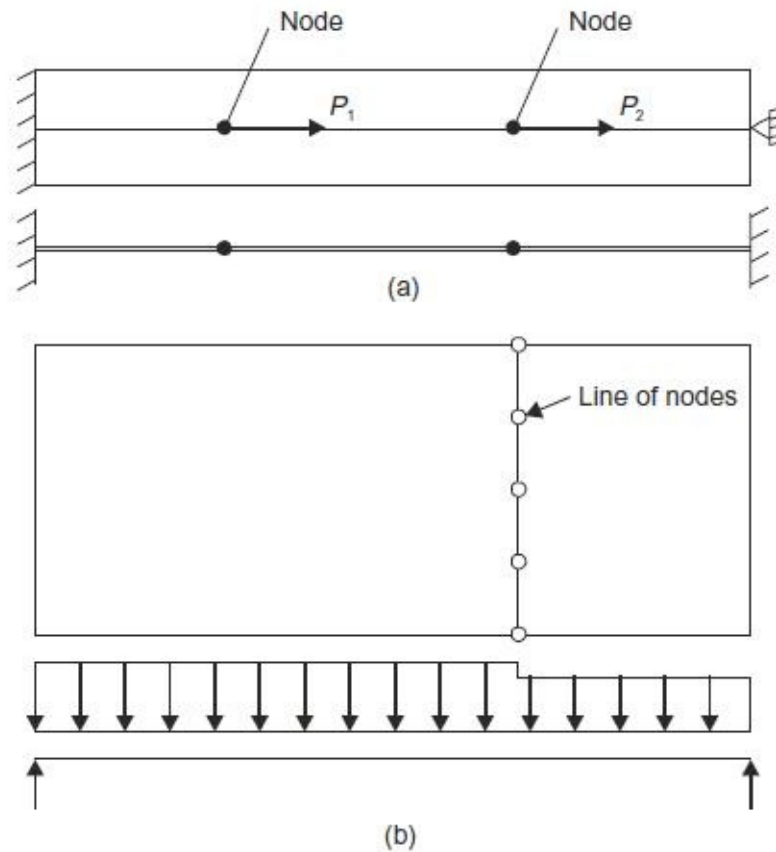


Fig. 10.2 (a) FEM model (b) Slab with different udl's

Discontinuity of Boundary conditions

If the boundary condition for a structure suddenly change we have to discretize such that there is node or a line of nodes. This type of situations are shown in Fig. 10.3.

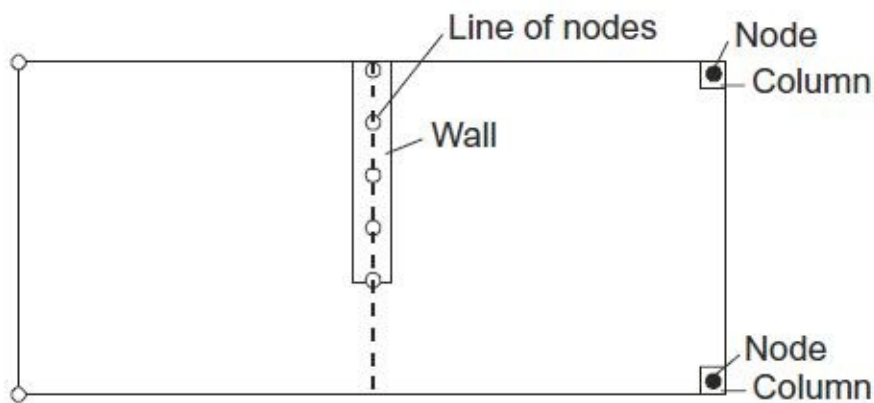


Fig. 10.3 Slab with intermediate wall and columns

Material Discontinuity

Node or node lines should appear at the places where material discontinuity is seen.

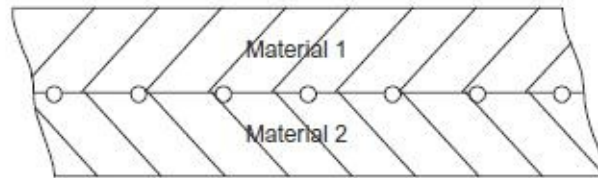


Fig. 10.4 *Material discontinuity*

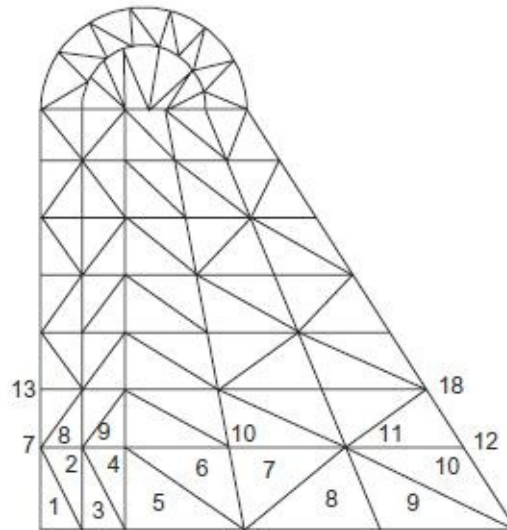


Fig. 10.5 *Refined mesh near curved boundary of a dam*

REFINING MESH

To get better results the finite element mesh should be refined in the following situations:

- (a) To approximate curved boundary of the structure
- (b) At the places of high stress gradients.

Such a situation is shown in Fig. 10.5

Use Of Symmetry

Wherever there is symmetry in the problem it should be made use. By doing so, lot of memory requirement is reduced or in other words we can use more elements (refined mesh) for the same capacity of computer memory. When symmetry is to be used, it is to be noted that at right angles to the line of symmetry displacement is zero.

HIGHER ORDER ELEMENTS V/S

REFINED MESH

Accuracy of calculation increases if higher order elements are used. Accuracy can also be

increased by using more number of elements. Limitation on use of number of elements comes from the total degrees of freedom the computer can handle. The limitation may be due to cost of computation time also. Hence to use higher order elements we have to use less number of such elements. The question arises whether to use less number of higher order elements or more number of lower order elements for the same total degree of freedom. There are some studies in this matter keeping degree of accuracy per unit cost as the selection criteria. However the cost of calculation is coming down so much that such studies are not relevant today. Accuracy alone should be selection criteria which may be carried out initially on the simplified problem and based on it element may be selection for detailed study.

OPTIMUM PROCESS OF FEA THROUGH SOLIDWORKS SIMULATION

With the knowledge of the basics discussed above we can summarize the process of performing analysis as follows:

1. Construct the part(s) in a solid modeler. It is surprisingly easy to accidentally build flawed models with tiny lines, tiny surfaces or tiny interior voids. The part will look fine, except with extreme zooms, but it may fail to mesh. Most systems have checking routines that can find and repair such problems before you move on to an FEA study. Sometimes you may have to export a part, and then import it back with a new name because imported parts are usually subjected to more time consuming checks than “native” parts. When multiple parts form an assembly, always mesh and study the individual parts before studying the assembly. Try to plan ahead and introduce split lines into the part to aid in mating assemblies and to locate load regions and restraint (or fixture or support) regions. Today, construction of a part is probably the most reliable stage of any study.

2. Defeature the solid part model for meshing. The solid part may contain features, like a raised logo, that are not necessary to manufacture the part, or required for an accurate analysis study. They can be omitted from the solid used in the analysis study. That is a relative easy operation supported by most solid modelers (such as the “suppress” option in SW) to help make smaller and faster meshes. However, it has the potential for introducing serious, if not fatal, errors in a following engineering study. This is a reliable modeling process, but its application requires engineering judgment. For example, removing small radius interior fillets can greatly reduces the number of elements and simplifies the mesh generation. But, that creates sharp reentrant corners that can yield false infinite stresses.

Those false high stress regions may cause you to overlook other areas of true high stress levels. Small holes lead to many small elements (and long run times). They also cause stress concentrations that raise the local stress levels by a factor of three or more. The decision to defeature them depends on where they are located in the part. If they lie in a high stress region you must keep them. But defeaturing them is allowed if you know they occur in a low stress region. Such decisions are complicated because most parts have

multiple possible loading conditions and a low stress region for one load case may become a high stress region for another load case.

3. Combine multiple parts into an assembly. Again, this is well automated and reliable from the geometric point of view and assemblies “look” as expected. However, geometric mating of part interfaces is very different for defining their physical (displacement, or temperature) mating. The physical mating choices are often unclear and the engineer may have to make a range of assumptions, study each, and determine the worst case result. Having to use physical contacts makes the linear problem require iterative solutions that take a long time to run and might fail to converge.

4. Select the element type. Some FEA systems have a huge number of available element types (with underlying theoretical restrictions). The SolidWorks system has only the fundamental types of elements. Namely, truss elements (bars), frame elements (beams), thin shells (or flat plates), thick shells, and solids. The SW simulation system selects the element type (beginning in 2009) based on the shape of the part. The user is allowed to convert a non-solid element region to a solid element region, and visa versa. Knowing which class of element will give a more accurate or faster solution requires training in finite element theory. At times a second element type study is used to help validate a study based on a different element type.

5. Mesh the part(s) or assembly, remembering that the mesh solid may not be the same as the part solid. A general rule in an FEA is that your computer never has enough speed or memory. Sooner or later you will find a study that you cannot execute. Often that means you must utilize a crude mesh (or at least crude in some region) and/or invoke the use of symmetry or antisymmetry conditions. Local solution errors in a study are proportional to the product of the local element size and the gradient of the secondary variables (i.e., gradient of stress or heat flux). Therefore, you exercise mesh control to place small elements where your engineering judgment estimates high stress (or flux) regions, as well as large elements in low stress regions.

The local solution error also depends on the relative sizes of adjacent elements. You do not want skinny elements adjacent to big ones. Thus, automatic mesh generators have options to gradually vary adjacent element sizes from smallest to biggest.

The solid model sent to the mesh generator frequently should have load or restraint (fixture) regions formed by split lines, even if such splits are not needed for manufacturing the parts. The mesh typically should have refinements at source or load regions and support regions.

A mesh must look like the part, but that is not sufficient for a correct study. A single layer

of elements filling a part region is almost never enough. If the region is curved, or subjected to bending, you want at least three layers of quadratic elements, but five is a desirable lower limit. For linear elements you at least double those numbers.

Most engineers do not have access to the source code of their automatic mesh generator. When the mesher fails you frequently do not know why it failed or what to do about it. Often you have to re-try the mesh generation with very large element sizes in hopes of getting some mesh results that can give hints as to why other attempts failed. The meshing of assemblies often fails. Usually the mesher runs out of memory because one or more parts had a very small, often unseen, feature that causes a huge number of tiny elements to be created. You should always attempt to mesh each individual part to spot such problems before you attempt to mesh them as a member of an assembly.

Automatic meshing, with mesh controls, is usually simple and fast today. However, it is only as reliable as the modified part or assembly supplied to it. Distorted elements usually do not develop in automatic mesh generators, due to empirical rules for avoiding them. However, distorted elements locations can usually be plotted. If they are in regions of low gradients you can usually accept them.

You should also note that studies involving natural frequencies are influenced most by the distribution of the mass of the part. Thus, they can still give accurate results with meshes that are much cruder than those that would be acceptable for stress or thermal studies.

6. Assign a linear material to each part. Modern FEA systems have a material library containing the “linear” mechanical, thermal, and/or fluid properties of most standardized materials. They also allow the user to define custom properties. The property values in such tables are often misinterpreted to be more accurate and reliable than they actually are. The reported property values are accepted average values taken from many tests. Rarely are there any data about the distribution of test results, or what standard deviation was associated with the tests. Most tests yield results that follow a “bell shaped” curve distribution, or a similar skewed curve.

When you accept a tabulated property value as a single number to be used in the FEA calculation remember it actually has a probability distribution associated with it. You need to assign a contribution to the total factor of safety to allow for variations from the tabulated property value.

The values of properties found in a material table can appear more or less accurate depending on the units selected. That is an illusion often caused by converting one set of units to another, but not truncating the result to the same number of significant figures available in the actual test units. For example, the elastic modulus of one steel is tabulated

from the original test as 210 MPa, but when displayed in other units it shows as 30,457,924.92 psi. Which one do you believe to be the experimental accurate value; the 3 digit value or the 10 digit one? The answer affects how you should view and report stress results. The axial stress in a bar is equal to the elastic modulus times the strain,

$$\sigma = E \varepsilon$$

. Thus, if E is only known to three or four significant figures then the reported stress result should have no more significant figures. Material data are usually more reliable than the loading values (considered

next), but less accurate than the model or mesh geometries.

7. Select regions of the part(s) to be loaded and assign load levels and load types to each region. In mathematical terminology, load or flux conditions on a boundary region are called Neumann boundary conditions, or nonessential conditions. The geometric regions can be points (in theory), lines, surfaces, or volumes. If they are not existing features of the part, then you should apply split lines to the part to create them before activating the mesh generator. Point forces, or heat sources, are common in undergraduate studies, but in an FEA they cause false infinite stresses, or heat flux. If you include them do not be misled by the high local values. Refining the mesh does not help since the smallest element still reports near infinite values.

In reality, point loads are better modeled as a total force, or pressure, acting over a small area formed by prior split lines. Saint Venant's Principle states that two different, but statically equivalent, force systems acting on a small portion of the surface of a body produce the same stress distributions at distance large in comparison with the linear dimensions of the portion where the forces act. That also implies that concentrated sources quickly become re-distributed.

In undergraduate statics and dynamics courses engineers are taught to think in terms of point forces and couples. Solid elements do not accept pure couples as loads, but statically equivalent pressures can be applied to solids and yield the correct stresses. Indeed, a couple at a point is almost impossible to create, so the distribution of pressures is probably more like the true situation.

The magnitudes of applied loads are often guesses, or specified by a governing design standard. For example, consider a wind load. A building standard may quote a pressure to be applied for a given wind speed. But, how well do you know the wind speed that might actually be exerted on the structure? Again, there probably is some type of "bell curve" around the expected average speed. You need to assign a contribution to the total factor of safety to allow for variations in the uncertainty of the load value or actual spatial distribution of applied loads.

Loading data are usually less accurate than the material data, but much more accurate than the restraint or supporting conditions considered next.

8. Determine (or more likely assume) how the model interacts with the surroundings not included in your model. These are the restraint (support, or fixture) regions. In mathematical terminology, these are called the essential boundary conditions, or Dirichlet boundary conditions. You cannot afford to model everything interacting with a part. For many

decades engineers have developed simplified concepts to approximate surroundings adjacent to a model to simplify hand calculations. They include roller supports, smooth pins, cantilevered (encastre, or fixed) supports, straight cable attachments, etc. Those concepts are often carried over to FEA approaches and can over simplify the true support nature and lead to very large errors in the results.

The choice of restraints (fixations, supports) for a model is surprisingly difficult and is often the least reliable decision made by the engineer. Small changes in the supports can cause large changes in the results. It is wise to try to investigate a number of likely or possible support conditions in different studies. When in doubt, try to include more of the surrounding support material and apply assumed support conditions to those portions at a greater distance from critical part features.

You need to assign a contribution to the total factor of safety to allow for variations in the uncertainty of how or where the actual support conditions occur. That is especially true for buckling studies.

9. Solve the linear system of equations, or the eigenvalue problem. With today's numerical algorithms the solution of the algebraic system or eigensystem is usually quite reliable. It is possible to cause ill-conditioned systems (large condition number) with meshes having bad aspect ratios, or large elements adjacent to small ones, but that is unlikely to happen with automatic mesh generators.

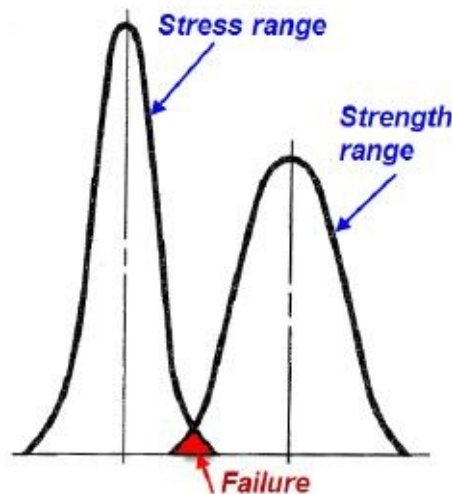
10. Check the results. Are the reactions at the supports equal and opposite to the sources you thought that you applied? Are the results consistent with the assumed linear behavior? The engineering definition of a problem with large displacements is one where the maximum displacement is more than half the smallest geometric thickness of the part. The internal definition is a displacement field that significantly changes the volume of an element.

That implies the element geometric shape noticeably changed from the starting shape, and that the shape needs to be updated in a series of much smaller shape changes. Are the

displacements big enough to require resolution with large displacement iterations turned on? Have you validated the results with an analytic approximation, or different type of finite element? Engineering judgments are required.

11. Post-process the solution for secondary variables. For structural studies you generally wish to document the deflections, reactions and stresses. For thermal studies you display the temperatures, heat flux vectors and reaction heat flows. With natural frequency models you show (or animate) a few mode shapes. In graphical displays, you can control the number of contours employed, as well as their maximum and minimum ranges. The latter is important if you want to compare two designs on a single page. Limit the number of digits shown on the contour scale to be consistent with the material modulus (or conductivity, etc.). Color contour plots often do not reproduce well, but graphs do, so learn to use them in your documentation.

12. Determine (or more likely assume) what failure criterion applies to your study. This stage involves assumptions about how a structural material might fail. There are a number of theories. Most are based on stress values or distortional energy levels, but a few depend on strain values. If you know that one has been accepted for your selected material then use that one (as a contribution to the overall factor of safety). Otherwise, you should evaluate more than one theory and see which is the worst case. Also keep in mind that loading or support uncertainties can lead to a range of stress levels, and variations in material properties affect the strength and unexpected failures can occur if those types of distributions happen to intersect as in the next figure.



14. Optionally, post-process the secondary variables to measure the theoretical error in the study, and adaptively correct the solution. This converges to an accurate solution to the problem input, but perhaps not to the problem to be solved. Accurate garbage is still garbage.

15. Document, report, and file the study. The part shape, mesh, and results should be reported in image form. Assumptions on which the study was based should be clearly stated, and hopefully confirmed. The documentation should contain an independent

validation calculation, or two, from an analytical approximation or an FEA based on a totally different element type.

PROJECT ON ANALYSIS

CHAPTER 14



Topics Covered

The major topics covered in this chapter are:

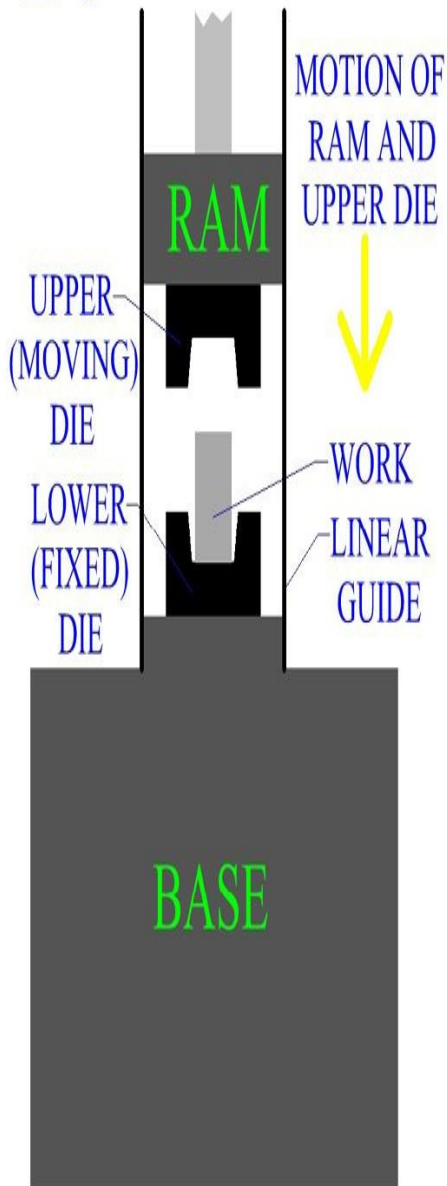
- *Project*
- *Static Analysis*
- *Frequency Analysis*
- *Fatigue Test*

PROJECT

In this project, we have a real-time problem of forging die. For your understanding, I want to tell you that there are many components like gears, shafts, rings, engine covers that are made with the help of a forging machine. In a forging machine, a workpiece pre-heated at temperature of plastic deformation is placed in the bottom die and top die strikes to the workpiece. After this impact of 50kN to 1600kN on the workpiece, the workpiece takes the shape of top and bottom dies; refer to [Figure-1](#).

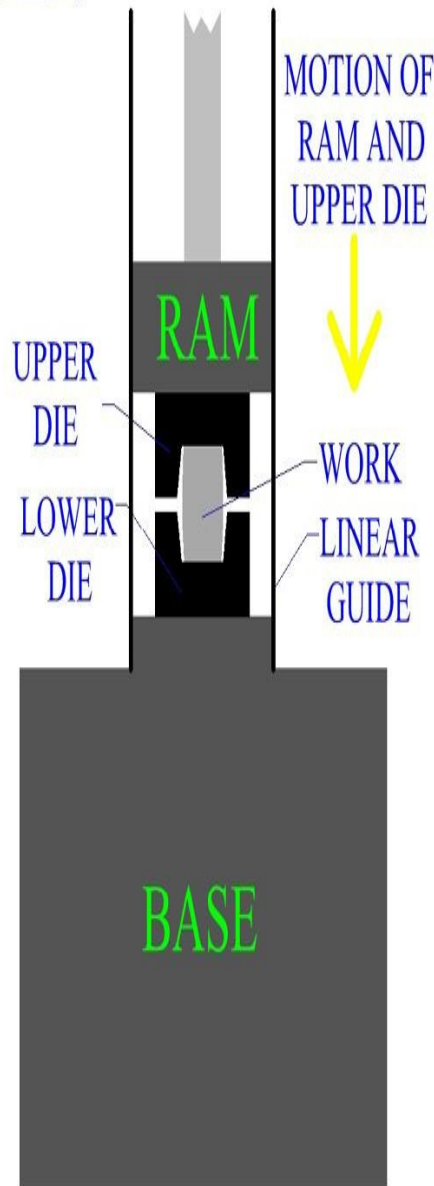
FORGING PRESS

1.



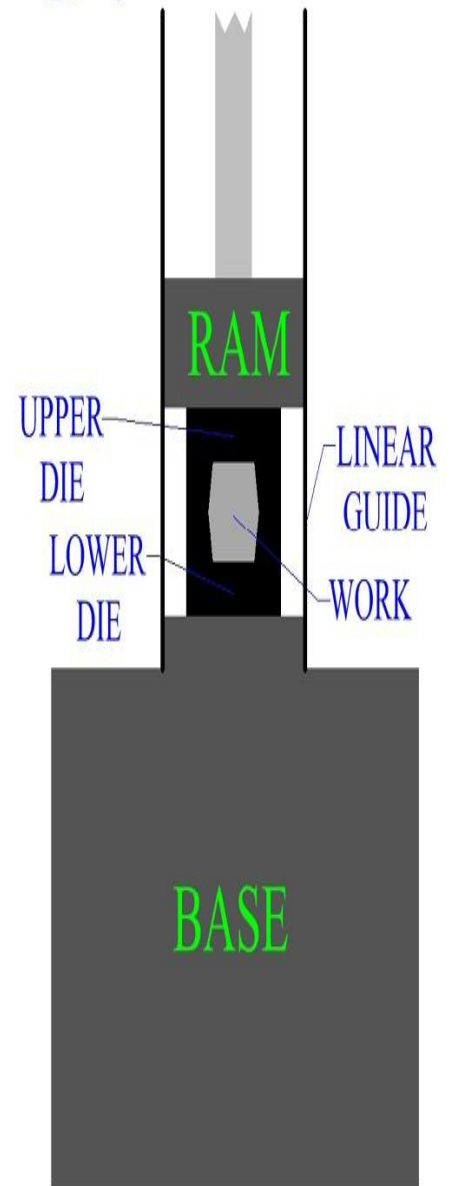
FORCE APPLIED FROM ABOVE CAUSES RAM AND UPPER DIE TO MOVE TOWARD WORK

2.



CONTINUOUS PRESSURE BEGINS TO FORM WORK

3.



THE CLOSING OF THE MOLD COMPLETES THE FORGING

Figure-1. Forging Process

Now, we have the bottom die that need to be analyzed at **115 °C** of surface temperature and force exerted on its surface is **200 kN**; refer to [Figure-2](#) and [Figure-3](#). We need to find out the Factor of safety of the model. If it is higher then methods to reduce the cost of model. If the FoS is lower then the methods to increase it for increasing strength of the model. Also, we need to check the model for Fatigue under the cycle of 100,000 loading/unloading. After performing the analysis, we will create a more refined mesh and then we will check the results again.

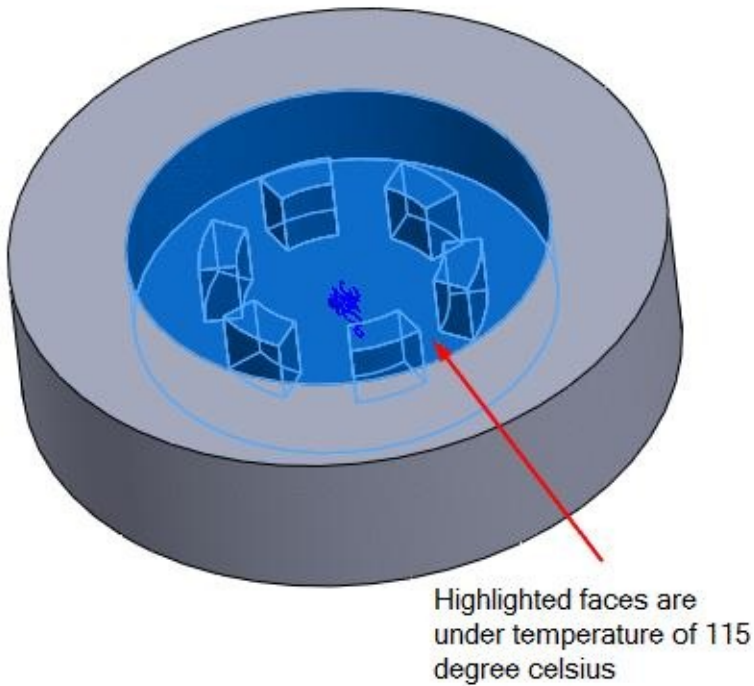
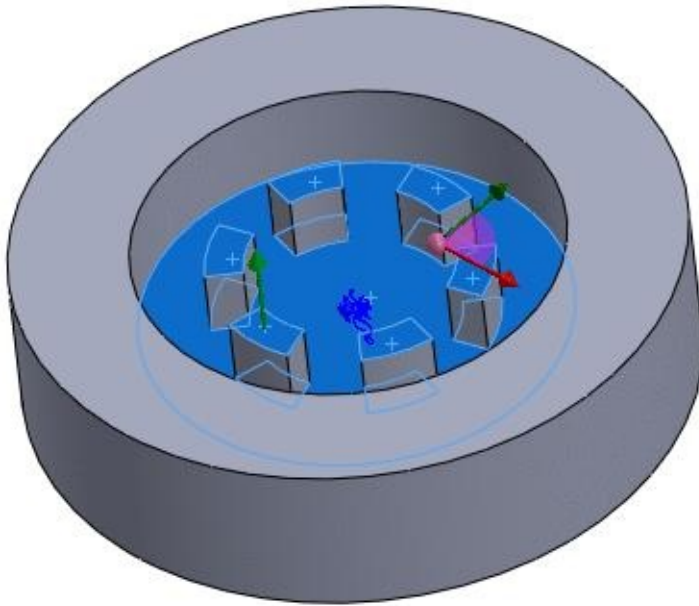


Figure-2. Faces under specified temperature



Highlighted faces are under the force of 200kN

Figure-3. Faces under the load

STARTING SIMULATION

- Open the file of die model in SolidWorks (The file is available in the Resource kit).
- Click on the **Simulation** button from the **SOLIDWORKS Add-Ins** tab in the Ribbon. The **Simulation** tab will be added in the **Ribbon**.
- Click on the **Simulation** tab. The tools related to simulation will be displayed.

PERFORMING THE STATIC ANALYSIS

- Click on the down arrow below **Study Advisor** in the **Simulation** tab's **Ribbon**.
- Select the **New Study** button from the list of tools displayed. The **Study PropertyManager** will be displayed.
- Click on the **Static** button from the **PropertyManager** and click on the **OK** button from the **PropertyManager**.
- Click on the **Apply Material** button from the **Ribbon**. The **Material** dialog box will be displayed as shown in [Figure-4](#).

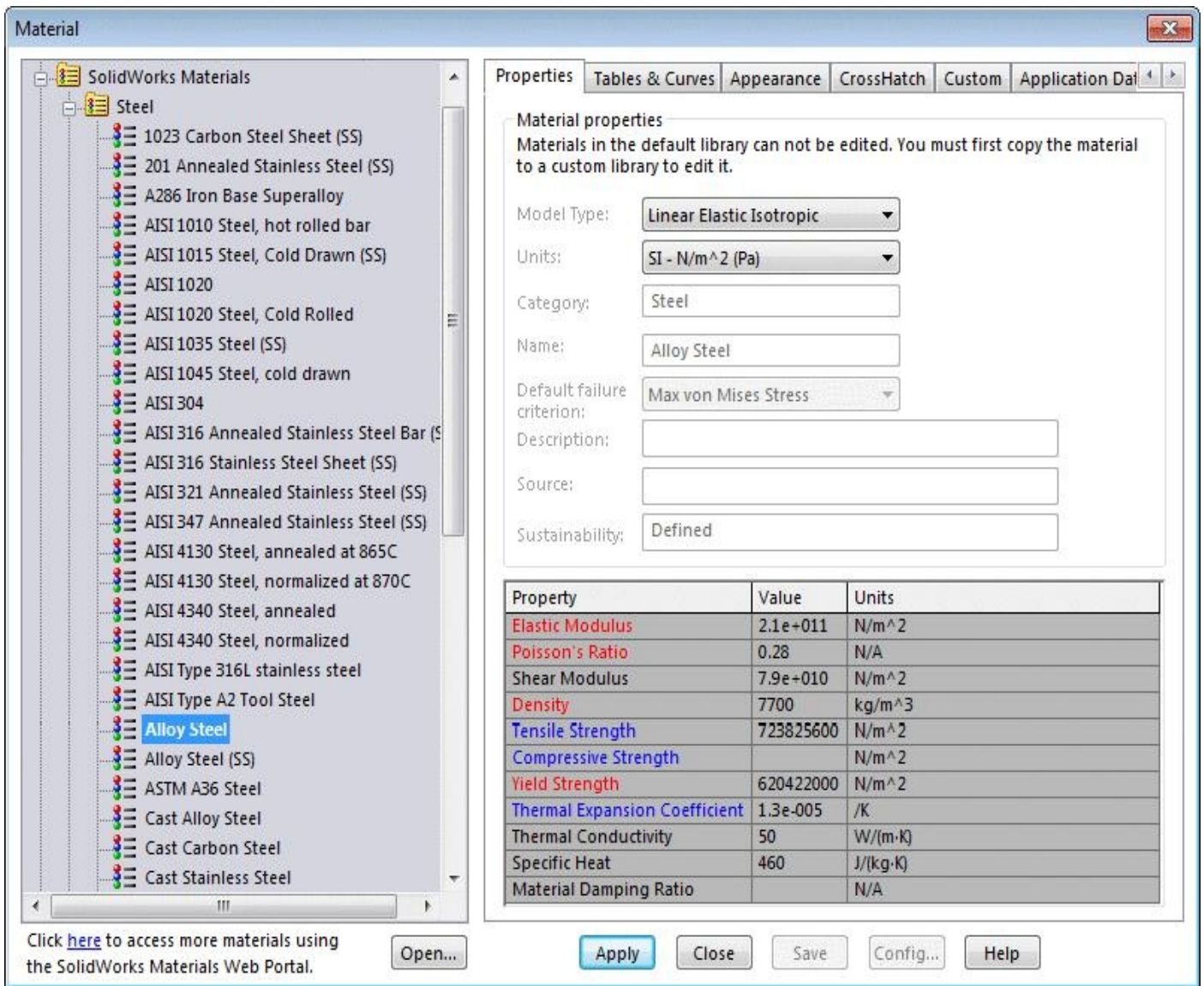


Figure-4. Material dialog box

- By default, the **Alloy Steel** material is selected from the list in the left of the dialog box. If not selected then select it.
- Click on the **Apply** button and then the **Close** button from the dialog box.
- Click on the down arrow below **Fixture Advisor** button in the **Ribbon** and select the **Fixed Geometry** button from the list.
- The **Fixture PropertyManager** will be displayed.
- Select the bottom face of the model to fix it; refer to [Figure-5](#).

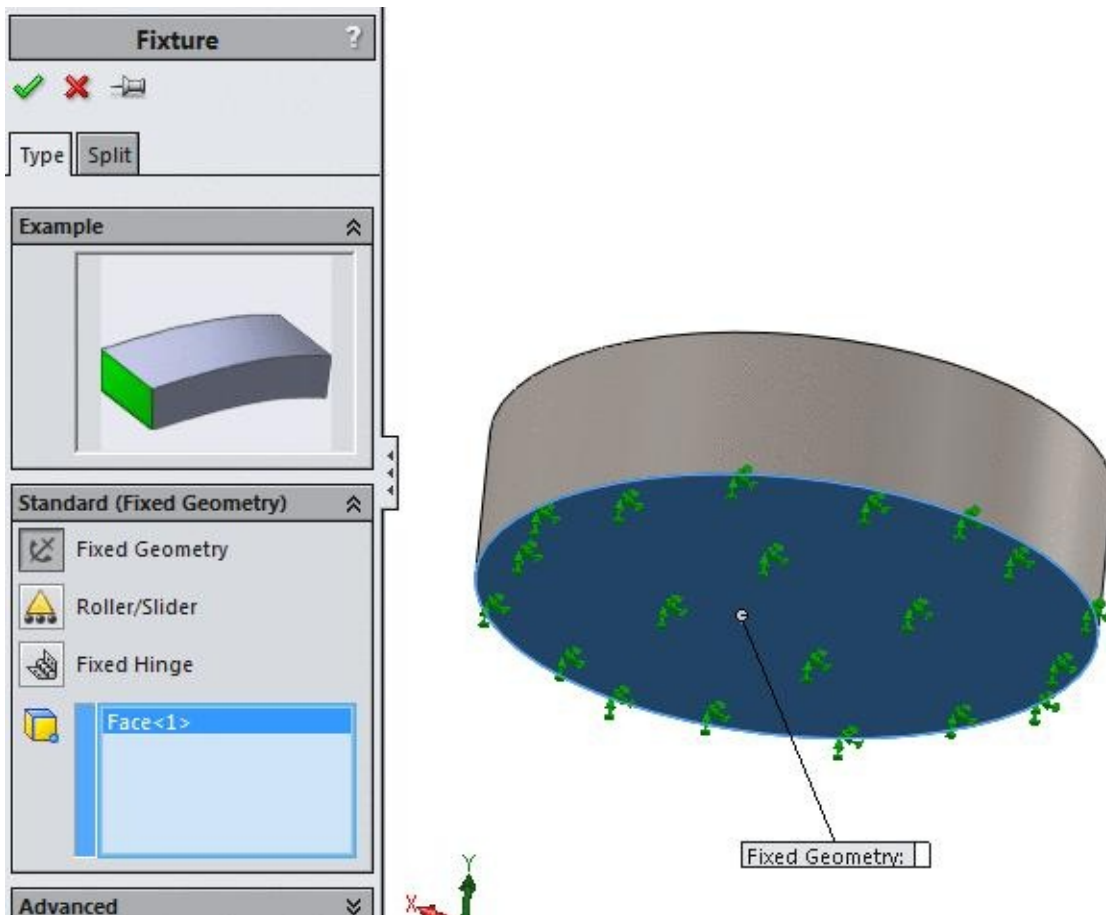


Figure-5. Face to be fixed

- Click on the **OK** button from the **PropertyManager**.
- Click on the down arrow below **External Loads Advisor** button in the **Ribbon**. The list of tools will be displayed.
- Click on the **Temperature** button from the list. The **Temperature PropertyManager** will be displayed.
- Click on the **Select all exposed faces** button from the **PropertyManager**. All the faces will get selected.
- Press and hold the **CTRL** key and select the bottom, side and top face of the die; refer to [Figure-6](#). The faces will get deselected.

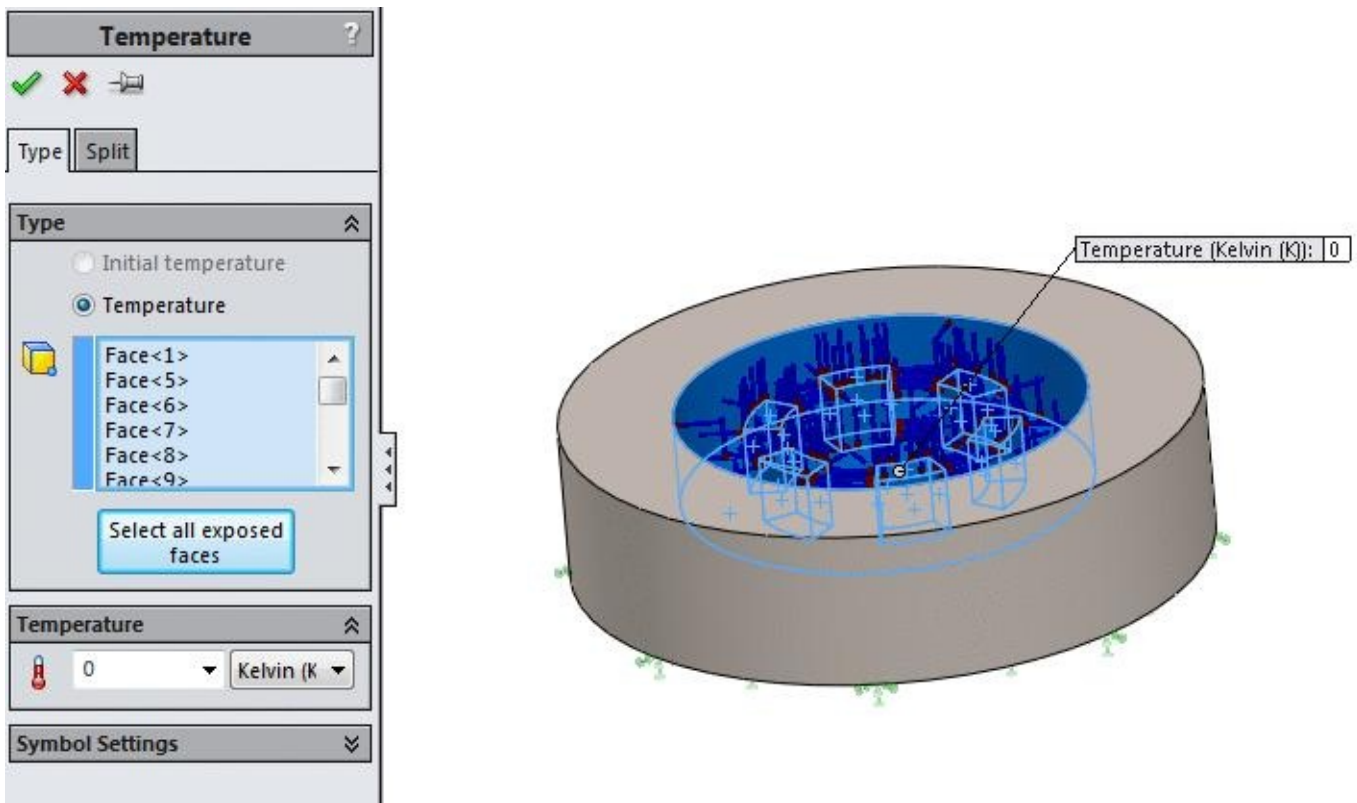


Figure-6. Faces selected for temperature

- Click on the **Temperature** drop-down in the **Temperature** rollout and select the **Celsius (°C)** option.
- Click in the **Temperature** edit box and specify the value as **115**.
- Click on the **OK** button from the **PropertyManager**.
- Again, click on the down arrow below **External Loads Advisor** button in the **Ribbon**.
- Select the **Force** button from the list of tools. The **Force PropertyManager** will be displayed.
- Select the faces visible from the top of the model; refer to [Figure-7](#).

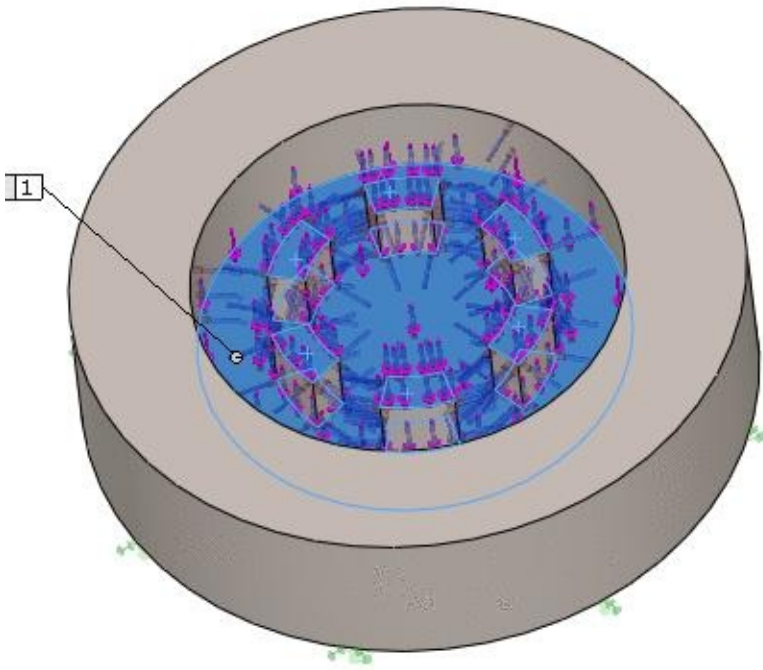


Figure-7. Faces for applying force

- Click in the **Force Value** edit box and specify the value as **200000N**.
- Select the **Total** radio button from the **PropertyManager** and click on the **OK** button.
- Click on the **Run** button from the **Ribbon**. After the analysis is solved the result will be displayed; refer to [Figure-8](#).

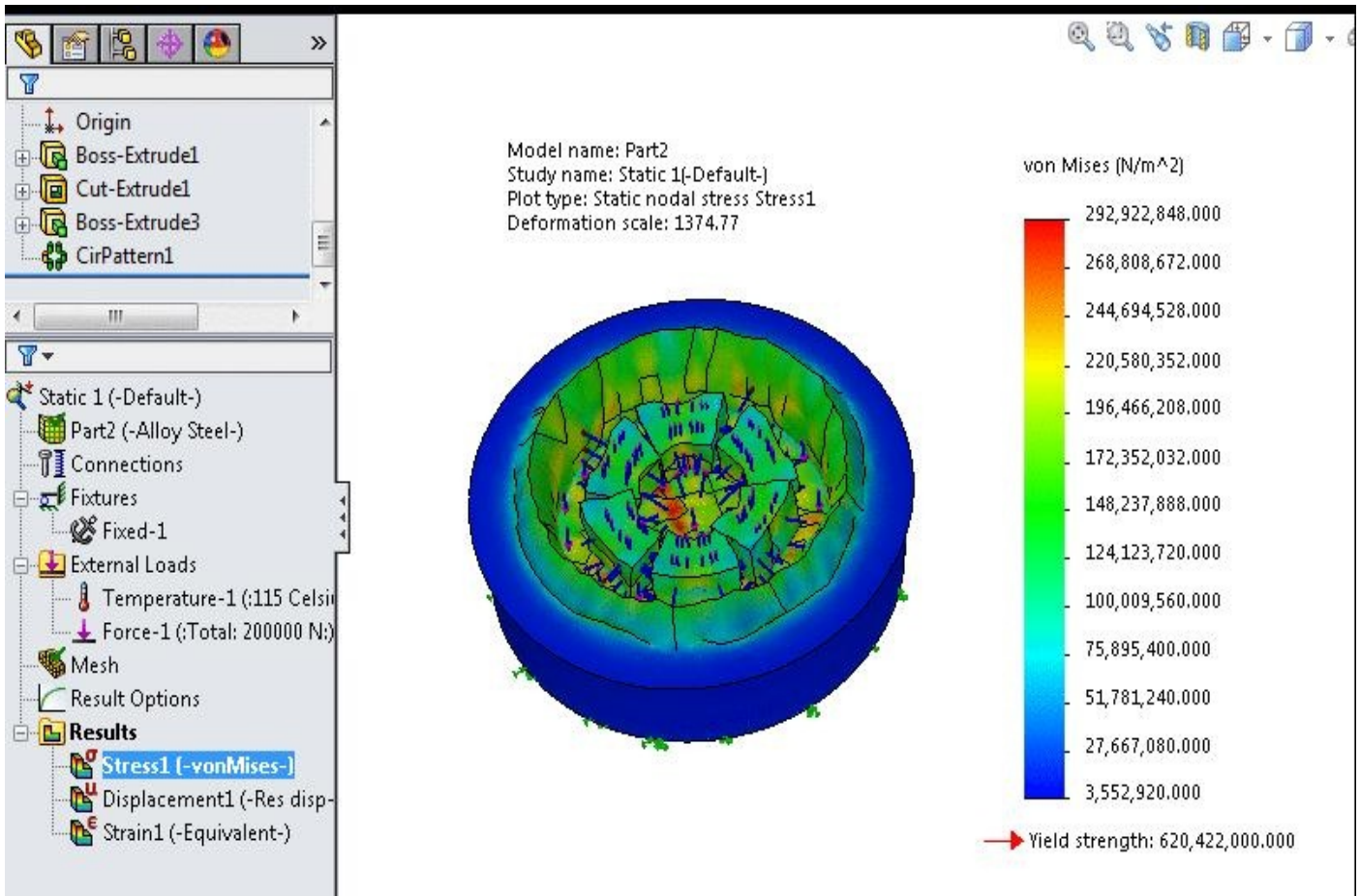


Figure-8. Result of static analysis

- Right-click on the **Results** node in Analysis Manager and select the **Define Factor Of Safety Plot** option from the shortcut menu displayed.
- The **Factor of Safety PropertyManager** will be displayed; refer to [Figure-9](#).

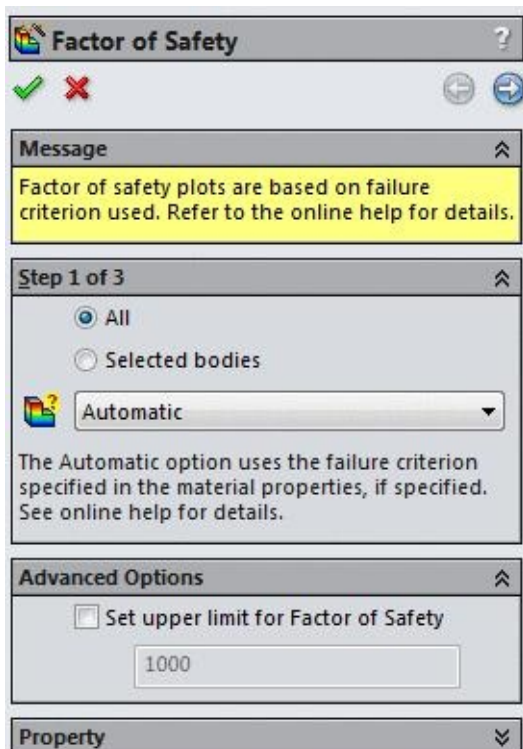


Figure-9. Factor of Safety PropertyManager

- Click on the **OK** button from the **PropertyManager**. The **Factor of Safety** plot will be displayed; refer to [Figure-10](#).

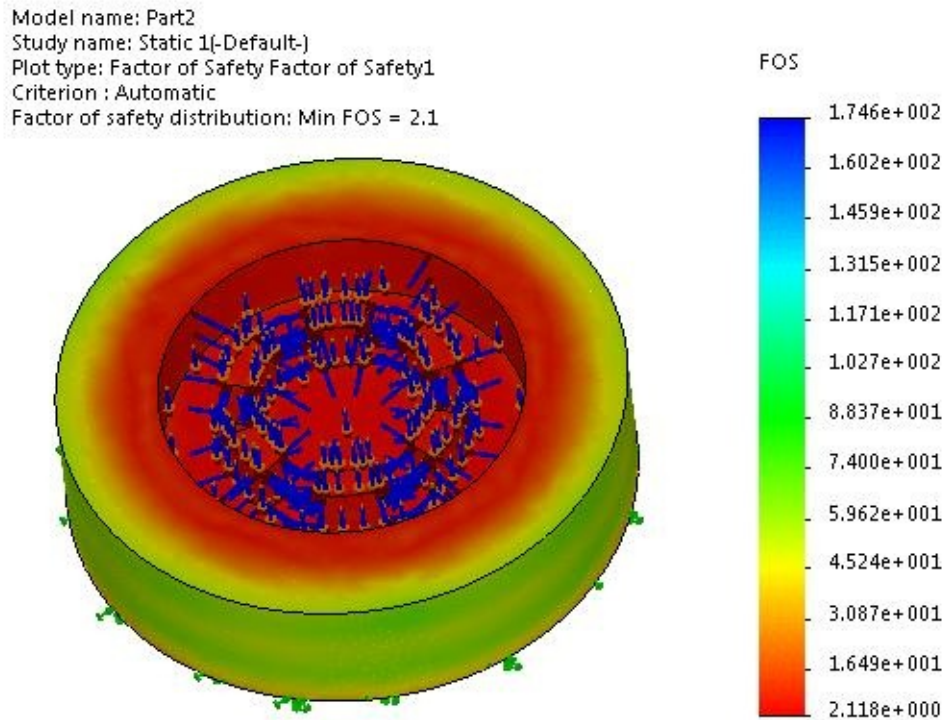


Figure-10. Factor of Safety plot

PERFORMING THE FREQUENCY ANALYSIS

- Click on the down arrow below **Study Advisor** and select the **New Study** button from the list displayed.
- Click on the **Frequency** button from the **Study PropertyManager** and click on the **OK** button from the **PropertyManager**. The tools related to frequency analysis will be displayed.
- Click on the **Apply Material** button from the **Ribbon**. The Material dialog box will display.
- Select the **Alloy Steel** from the left of the dialog box.
- Click on the **Apply** button and then on the **Close** button from the dialog box.
- Fix the bottom face of the die as done in the previous analysis by using the **Fixed Geometry** button.
- Click on the **Run** button from the **Ribbon** to perform the analysis.
- After the analysis is complete, the natural frequencies of the model will be displayed;

refer to [Figure-11](#).

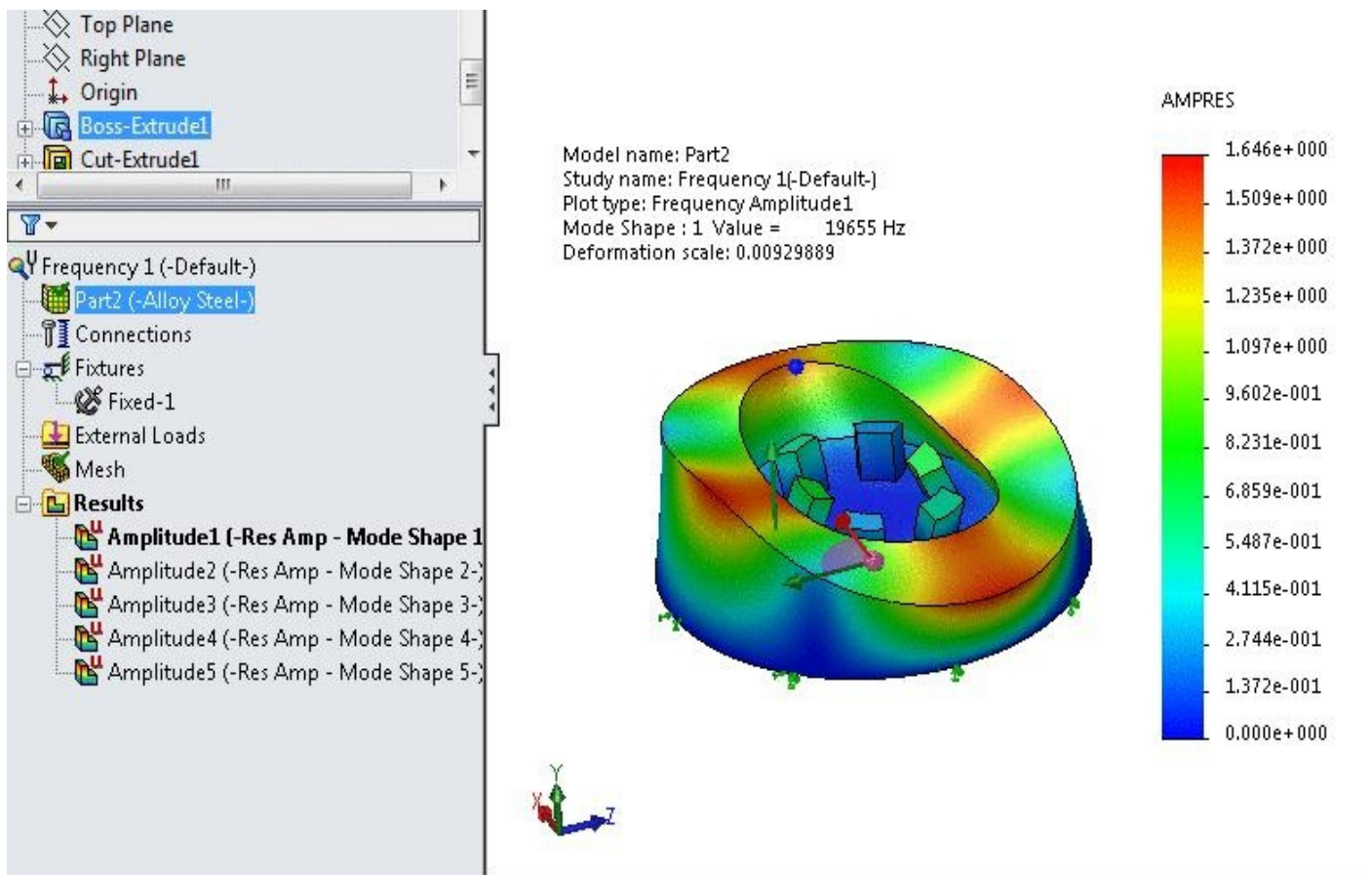


Figure-11. Result of frequency analysis

- Make sure that the frequency of the machine vibration is not matching with the natural frequencies of the model.

PERFORMING FATIGUE STUDY

- Click on the down arrow below **Study Advisor** in the **Ribbon**. List of the tools will be displayed.
- Click on the **New Study** tool. The **Study PropertyManager** will be displayed.
- Click on the **Fatigue** button from the **PropertyManager** and click on the **OK** button from the **PropertyManager**. The tools related to **Fatigue analysis** will be displayed.
- Right-click on **Loading (-Constant Amplitude-)** in the **Analysis Manager** and select the **Add Event** option; refer to [Figure-12](#). The **Add Event (Constant) PropertyManager** will be displayed; refer to [Figure-13](#).

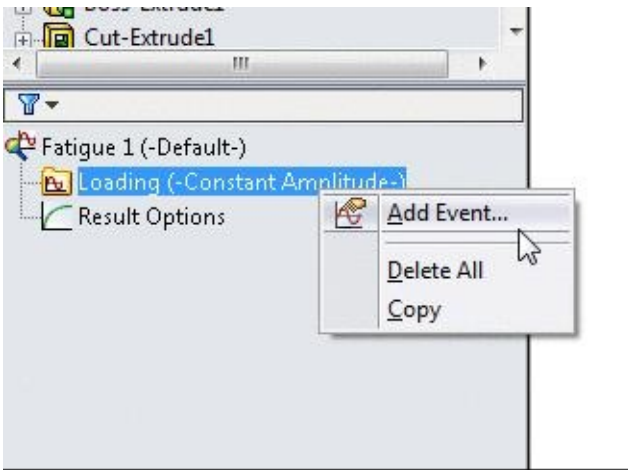


Figure-12. Add Event option

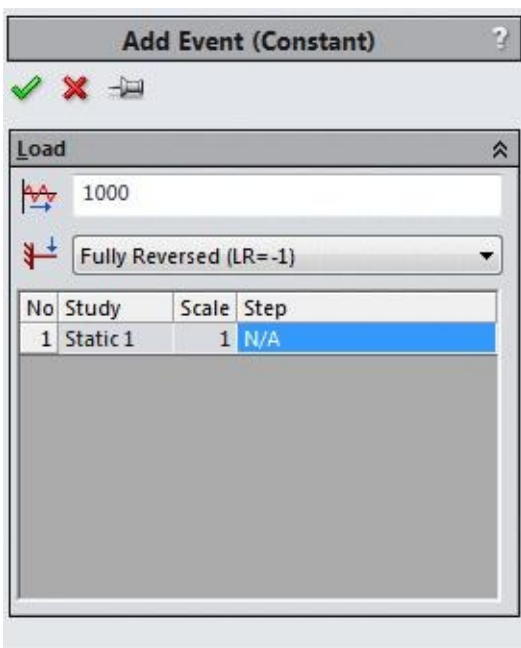


Figure-13. Add Event PropertyManager

- Click in the **Cycles** edit box and specify the value as **100,000**.
- Click on the **Loading Type** drop-down and select the **Zero Based (LR=0)** option from the drop-down list.
- Click on the **OK** button from the **PropertyManager** to add the event. Note that name of part will be added as a new node in the **Analysis Manager**.
- Right-click on the name of the part and select the **Apply/Edit Fatigue Data** option; refer to [Figure-14](#). The **Material** dialog box will be displayed with the **Fatigue SN Curves** tab activated; refer to [Figure-15](#).

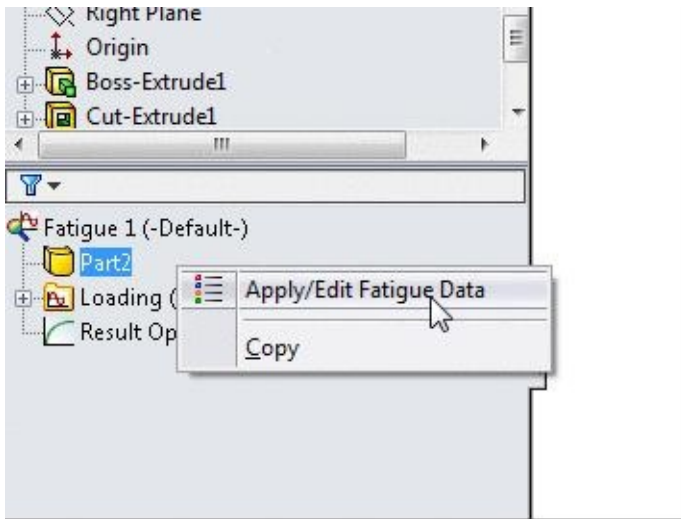


Figure-14. Applying fatigue data

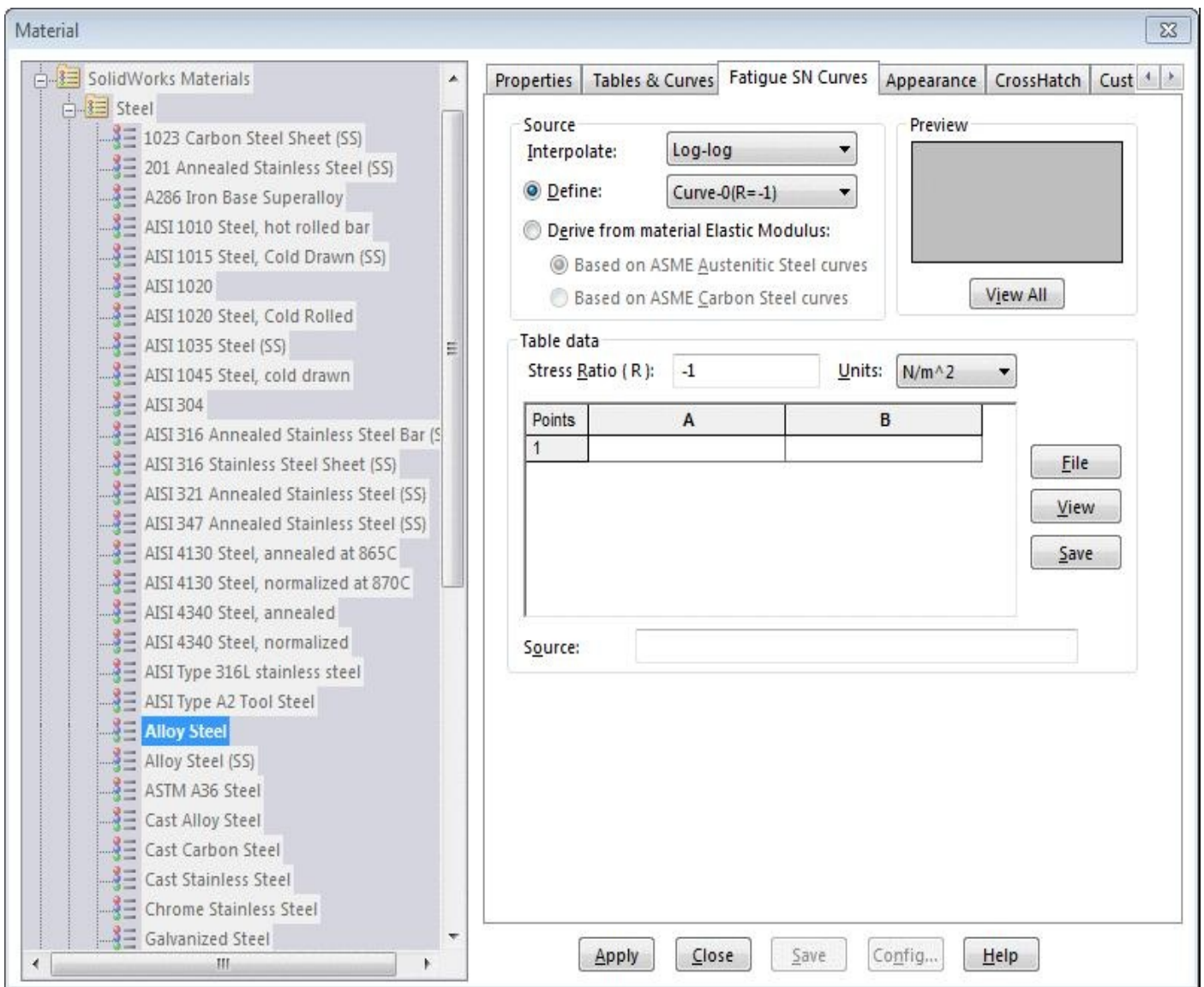


Figure-15. Material dialog box with fatigue options

- Click on the **Derive from material Elastic Modulus** radio button and then click on the **Based in ASME Carbon Steel curves** radio button. (Due to selection of these radio

buttons, the SN curve of a carbon steel will be used which is derived from its elasticity curve.)

- Click on the **Apply** button and then click on the **Close** button from the dialog box. Now, we are ready to perform the analysis.
- Click on the **Run** button from the **Ribbon**. The analyzing will start.
- The results of the analysis will be added in the Results node.
- Expand the node and double-click on the results to display the plot; refer to [Figure-16](#).

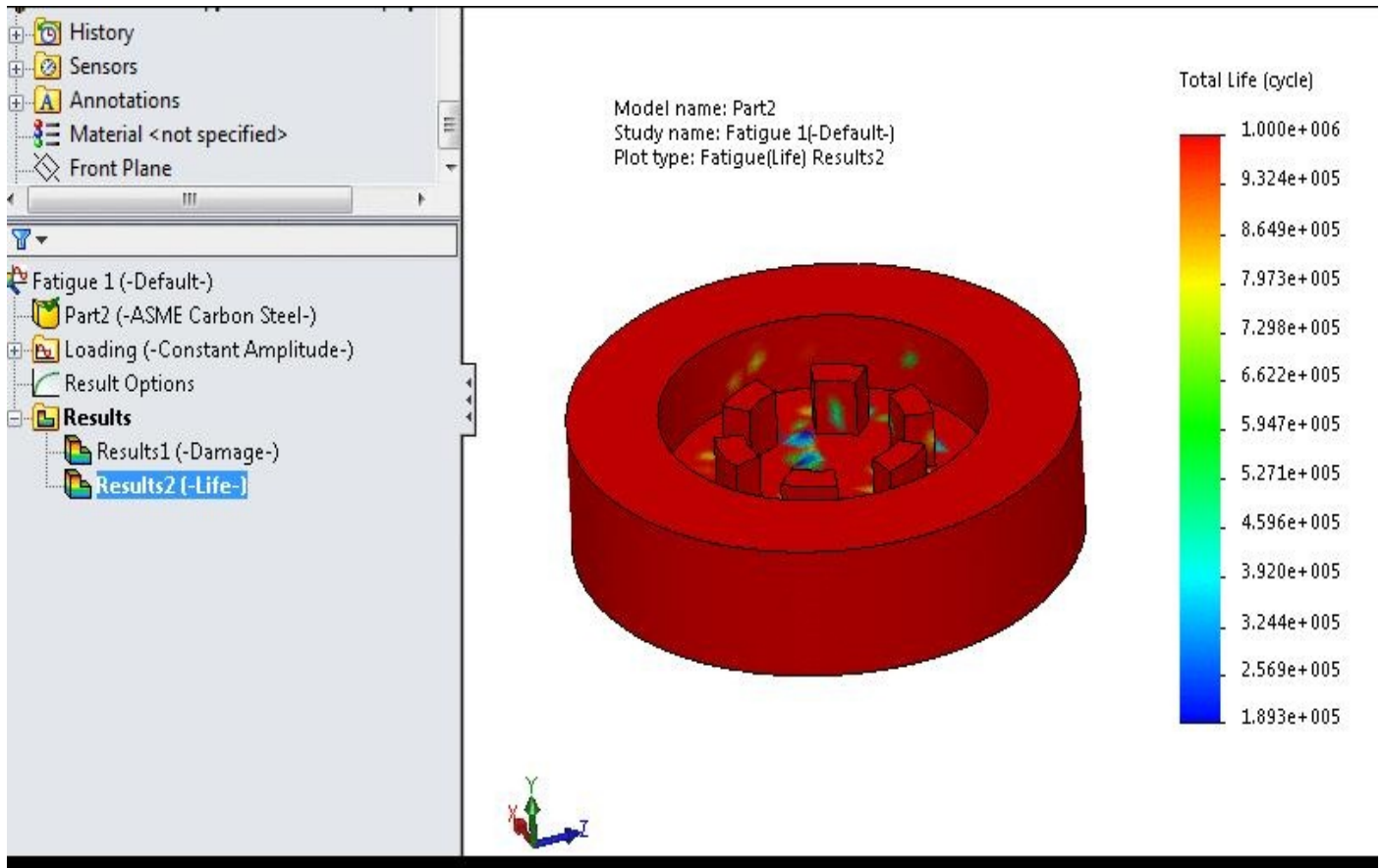


Figure-16. Fatigue result

- Right-click on the **Results** node and select the **Define Fatigue Plot** option; refer to [Figure-17](#). The **Fatigue Plot PropertyManager** will be displayed; refer to [Figure-18](#).

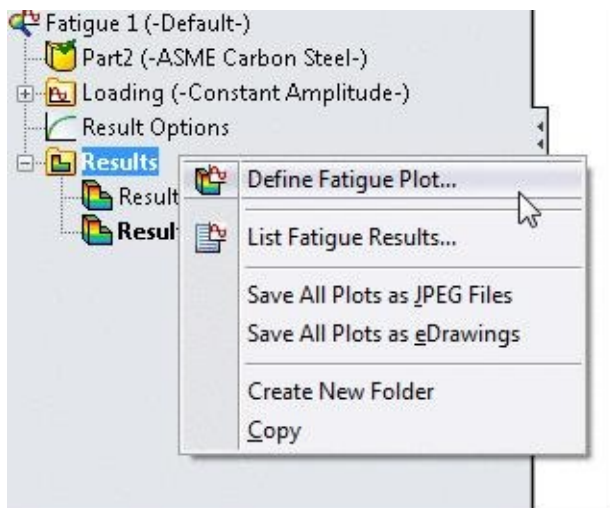


Figure-17. Define fatigue plot option

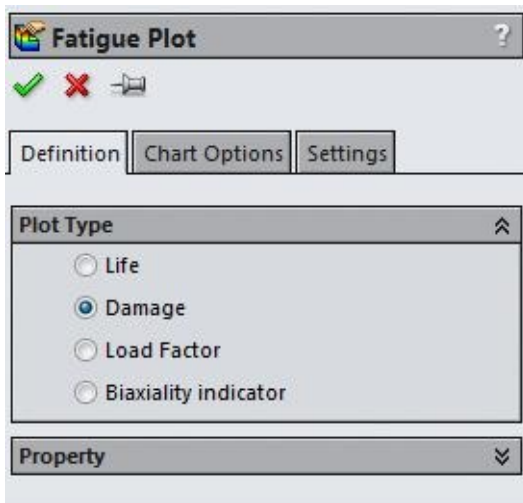


Figure-18. Fatigue Plot PropertyManager

- Click on the **Load Factor** radio button and click on the **OK** button from the **PropertyManager**. The load factor plot (in other words, **Factor of Safety** for fatigue) will be displayed; refer to [Figure-19](#).

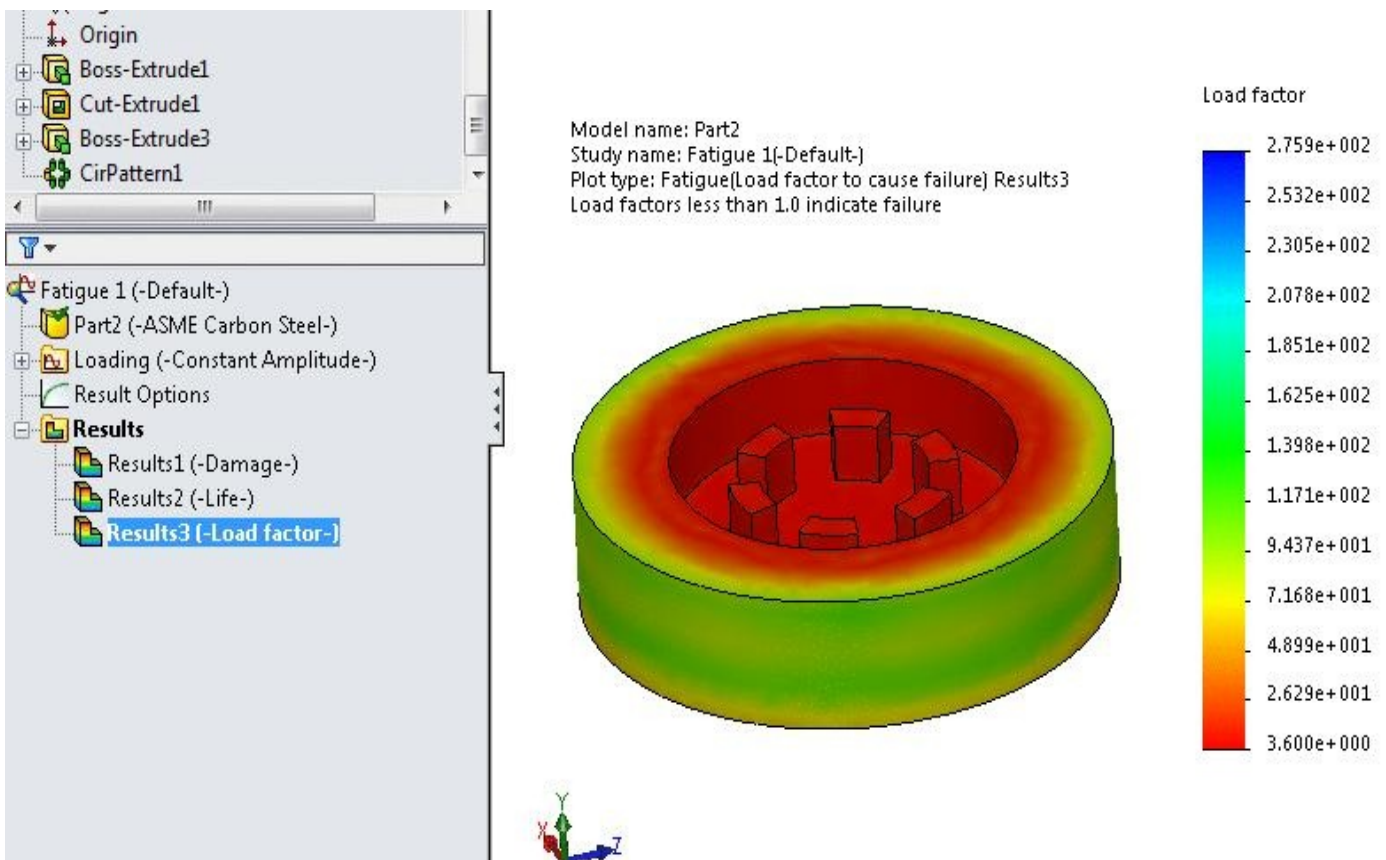


Figure-19. Factor of safety fatigue

- From the scale, you can find that the minimum factor of safety of the fatigue study is **3.6**.

Since our **Factor of Safety** for the problem is optimum, so we do not required to perform the **Design Study** as performed in previous chapters. Although, you can perform the Design Study to find a better product.

Now, I have a little work for you:

Go back to **Static Study** analysis by clicking on the **Static 1** tab at the bottom bar of the modeling area. Note that the **Factor of Safety** is **2.1** for this analysis. Now, Right-click on the **Mesh** node in the **Analysis Manager**; refer to [Figure-20](#).

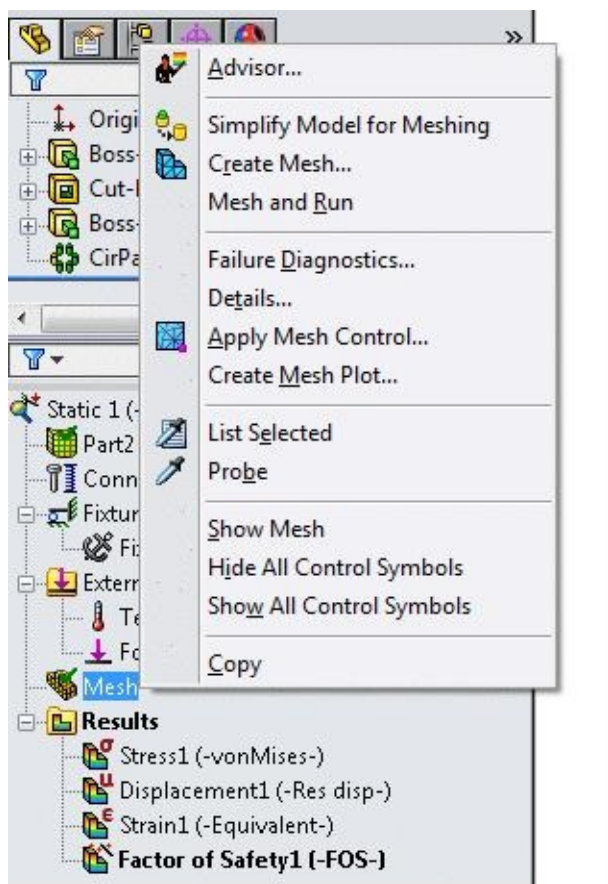


Figure-20. Shortcut menu for Mesh

Select the **Create Mesh** button, make the meshing extremely fine and then re-run the static analysis. Now, check the **Factor of Safety** of the model. Is it the same or different.

If different then why? Ask your tutor.

PRACTICE QUESTIONS



Topics Covered

The major topics covered in this chapter are:

- *Problems for practice*

PROBLEM 1

Sometimes we see people fighting and in some extreme cases, fighting with the help of Baseball bats. We are not here to discuss concepts of fighting but once the fighting is over, everybody is busy to see effect of baseball bat on humans. Due to unknown elastic property of human body, we can not perform analysis to find effect of baseball bat on humans, but here we can check the effect of force exerted on the baseball bat due to some freaks; refer to [Figure-1](#).



Figure-1. Problem 1

We assume it to be a static analysis. Check whether the bat will break or not.

PROBLEM 2

The wall of a house is 7 m wide, 6 m high and 0.3 m thick made up of brick. The thermal conductivity of brick is $k = 0.6 \text{ W/m.K}$. If the inside temperature of wall is $16 \text{ }^\circ\text{C}$ and outside temperature is $6 \text{ }^\circ\text{C}$ then find out the heat lost from inside wall to outside wall in a steady state condition. The model is given in [Figure-2](#).

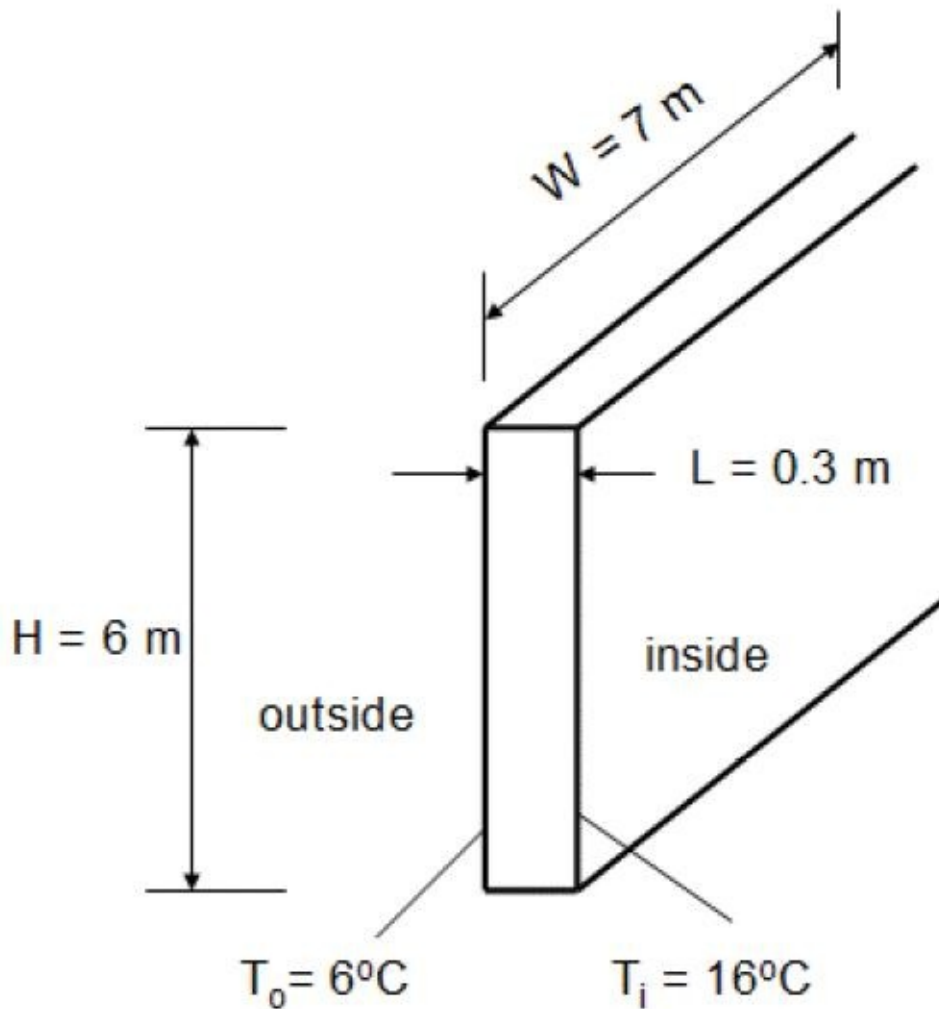


Figure-2. Problem 2

Note that to perform this thermal analysis, you need to create a new material with thermal properties as follow:

Thermal Conductivity = $0.6 \text{ W}/(\text{m}\cdot\text{K})$

Specific heat = $1214 \text{ J}/(\text{kg}\cdot\text{K})$

Mass density = $2405 \text{ kg}/\text{m}^3$

PROBLEM 3

An axial pull of **35kN** is applied at the ends of a shaft as shown in [Figure-3](#). If the material is **Alloy Steel** then find the total elongation in the shaft.

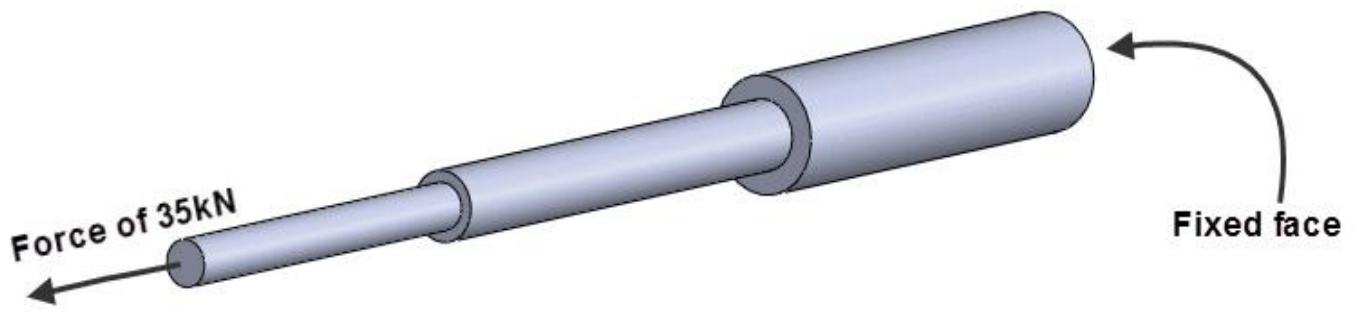


Figure-3. Problem 3 shaft