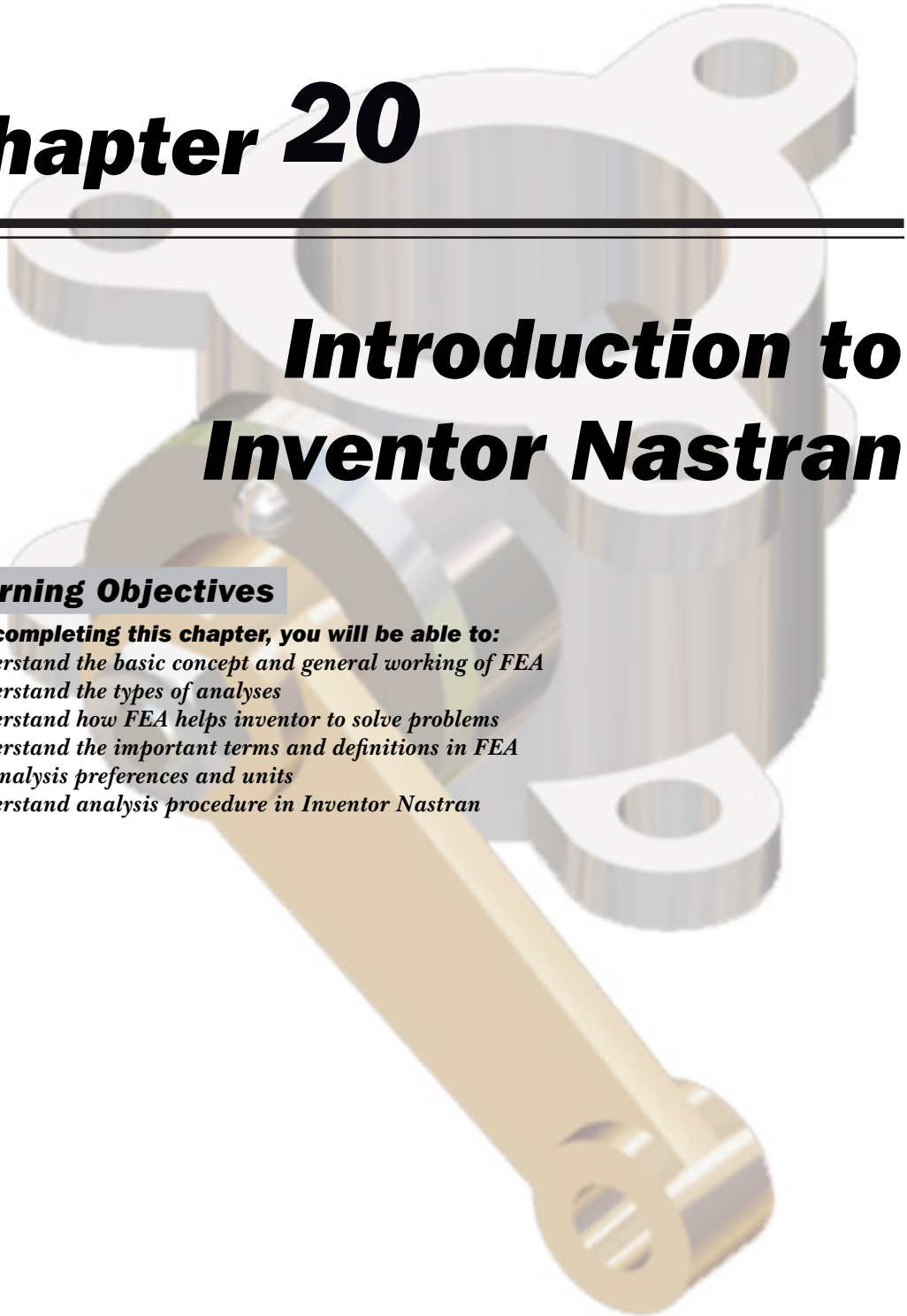
Chapter 20

Introduction to Inventor Nastran



Learning Objectives

After completing this chapter, you will be able to:

- *Understand the basic concept and general working of FEA*
- *Understand the types of analyses*
- *Understand how FEA helps inventor to solve problems*
- *Understand the important terms and definitions in FEA*
- *Set analysis preferences and units*
- *Understand analysis procedure in Inventor Nastran*

INTRODUCTION TO INVENTOR NASTRAN

Autodesk Inventor Nastran is an FEA tool embedded in Autodesk Inventor as an add-in. Designed specifically as a tool for predicting the physical behavior of practically any part or assembly under various boundary conditions, it offers simulation capabilities spanning across multiple analysis types such as linear and nonlinear stress, dynamics, and heat transfer. These analyses are mathematical models that has differential equations with a set of boundary values and initial conditions for a physical situation. The method used to derive these equations is called Finite Element Method (FEM). Inventor Nastran is available as a network license and serves multiple CAD platforms, to provide a consistent user experience and eliminate the need for multiple single-platform simulation technologies. It delivers high-end simulation technology in a CAD-embedded workflow so that you can make great products.

Apart from analyzing conventional engineering materials, Autodesk Inventor Nastran offers solution capabilities such as virtual simulation of structural, thermal, dynamics, and nonlinear problems; simulation of the latest advanced materials including composites, plastic, nonlinear elastic, shape memory, hyperelastic, viscoelastic; and simulation of brittle materials such as concrete.

TYPES OF ANALYSIS

The following types of analysis can be performed by using the FEA software:

1. Structural analysis
2. Thermal analysis
3. Fluid flow analysis
4. Electromagnetic field analysis
5. Coupled field analysis

Structural Analysis

In structural analysis, first the nodal degrees of freedom (displacement) are calculated and then the stress, strains, and reaction forces are calculated from nodal displacements. The classification of structural analysis is shown in Figure 20-1. The types of structural analysis are discussed next.

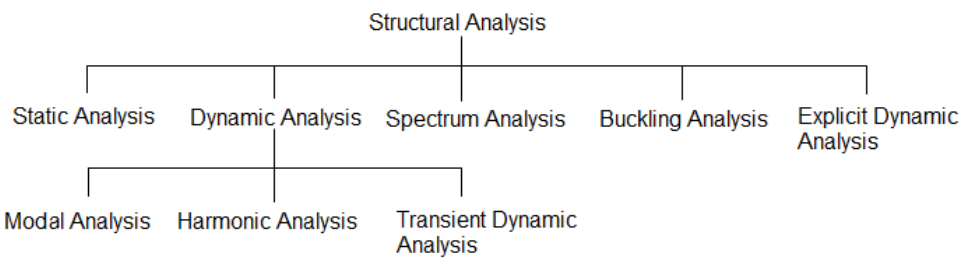


Figure 20-1 *Types of structural analysis*

Static Analysis

In static analysis, the load or field conditions do not vary with respect to time, and therefore, it is assumed that the load or field conditions are applied gradually, not suddenly. The system under this analysis can be linear or nonlinear. The inertia and damping effects are ignored in structural analysis. In structural analysis, the following matrices are solved:

$$[K] \times [X] = [F]$$

Where,

K = Stiffness Matrix
X = Displacement Matrix
F = Load Matrix

The above equation is called the force balance equation for the linear system. If the elements of matrix [K] are the function of [X], the system is known as the nonlinear system. Nonlinear systems include large deformation, plasticity, creep, and so on. The loadings that can be applied in a static analysis include:

1. Externally applied forces and pressures
2. Steady-state inertial forces (such as gravity or rotational velocity)
3. Imposed (non-zero) displacements
4. Temperatures (for thermal strain)
5. Fluences (for nuclear swelling)

Following can be the outputs of analysis performed by FEA software:

1. Displacements
2. Strains
3. Stresses
4. Reaction forces

Dynamic Analysis

In dynamic analysis, the load or field conditions do vary with time. In this analysis, the assumption is that the load or field conditions are applied suddenly. The system can be linear or nonlinear. The dynamic load includes oscillating loads, impacts, collisions, and random loads. The three main categories of the dynamic analysis are discussed next.

Modal Analysis

It is used to calculate the natural frequency and mode shape of a structure.

Harmonic Analysis

It is used to calculate the response of a structure to the loads that are varying with time harmonically.

Transient Dynamic Analysis

It is used to calculate the response of a structure to arbitrary time varying loads.

Spectrum Analysis

This is an extension of the modal analysis and is used to calculate stress and strain due to the response of the spectrum (random vibrations). For example, you can use it to analyze how well a structure will perform and survive in an earthquake.

Buckling Analysis

This type of analysis is used to calculate the buckling load and the buckling mode shape. Slender structures (thin and long) when loaded in the axial direction, buckle under relatively small loads. For such structures, the buckling load becomes a critical design factor.

Explicit Dynamic Analysis

This type of structural analysis is used to get fast solutions for large deformation dynamics and complex contact problems. For example, explosions, aircraft crash worthiness, and so on.

Thermal Analysis

The thermal analysis is used to determine the temperature distribution and related thermal properties such as:

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

All primary heat transfer modes such as conduction, convection, and radiation can be simulated. You can perform two types of thermal analysis by using the FEA software: Steady-State and Transient.

Steady State Thermal Analysis

In this analysis, the system is studied under steady thermal loads with respect to time.

Transient Thermal Analysis

In this analysis, the system is studied under varying thermal loads with respect to time.

Fluid Flow Analysis

This analysis is used to determine the flow distribution and temperature of a fluid. The outputs that are expected from the fluid flow analysis are velocities, pressure, temperature, and film coefficients.

Electromagnetic Field Analysis

This type of analysis is conducted to determine the magnetic fields in electromagnetic devices. The types of electromagnetic analyses are:

1. Static analysis
2. Harmonic analysis
3. Transient analysis

Coupled Field Analysis

This type of analysis considers the mutual interaction between multiple fields. It is impossible to solve fields separately because they are interdependent. Therefore, you need a program that can solve both the physical problems by combining them.

For example, if a component is bent in different shapes using one of the metal forming processes and then subjected to heating, the thermal characteristics of the component will depend on the new shape of the component. Therefore, first the shape of the component has to be determined through structural simulations. This is known as coupled field analysis.

SIMULATION OF DIGITAL PROTOTYPES

1. You can validate the product designs to learn how your parts will perform.
2. It allows you to explore design options at the beginning of the design cycle.
3. With simulation, you can predict product performance before creating and testing expensive prototypes.

WHY DO WE NEED FEA?

1. Predict performance for planned use.
2. Evaluate and correct the observed failure.
3. Predict possible failures with predictable abuse.
4. Improve the performance and security of a familiar design.
5. Develop new and innovative concepts.
6. Get information on design concepts or instructions.
7. Modeling decisions must be guided by objectives.

GENERAL PROCEDURE TO CONDUCT FINITE ELEMENT ANALYSIS USING INVENTOR NASTRAN

The following steps are used to conduct the Finite Element Analysis:

1. Create the model
2. Assign the material
3. Idealizations of model
4. Apply constraints
5. Create mesh of the model
6. Apply loads
7. Review the result

FEA through Software

The Finite Element Analysis process can be carried out in three main phases using software: preprocessor, solution, and postprocessor, refer to Figure 20-2.

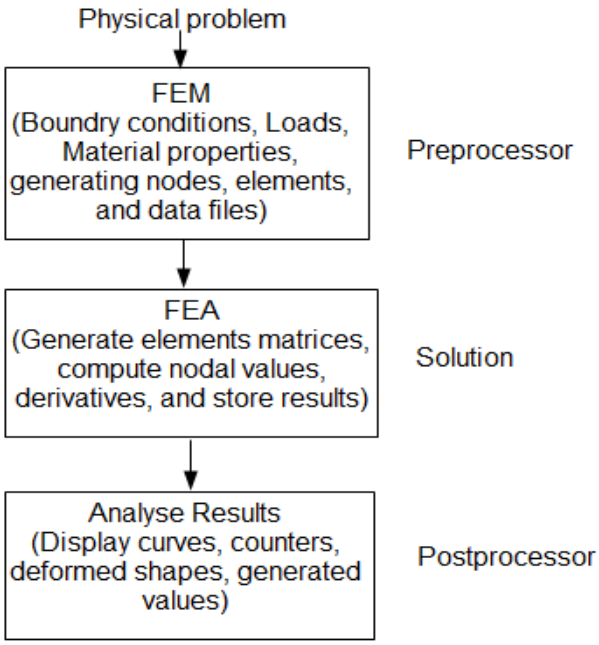


Figure 20-2 *The general process of FEA*

Preprocessor

The preprocessor is a phase that processes the input data to produce the results, which are used as input in the subsequent phase (solution). The following are the input data required in this phase:

1. Type of analysis
2. Geometric Model
3. Meshing
4. Material properties
5. Applying constraints
6. Loadings and boundary conditions

The input data are preprocessed for the output data. These data files are used in the subsequent phase (solution), refer to Figure 20-2.

Solution

The solution phase is completely automatic. The FEA software generates element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyze the results through the graphic display and tabular listings, refer to Figure 20-2.

Postprocessor

The output from the solution phase (result data files) is in numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output of the postprocessor is nodal displacement and stress in elements. The postprocessor processes the result data and displays them in graphical form to check or analyze the result. The graphical output gives the detailed information about the required result data. The postprocessor phase is automatic and generates graphical output in the specified form, refer to Figure 20-2

IMPORTANT TERMS AND DEFINITIONS

There are some important terms and definitions used in FEA software that are discussed next.

Strength

When a material is subjected to an external load, the system undergoes a deformation. The material in turn offers resistance against this deformation. This resistance is offered by the material by virtue of its strength.

Load

The external force acting on a body is called load.

Stress

The force of resistance offered by a body per unit area against the deformation is called stress. The stress is induced in the body due to the load is being applied on the body. The stress is calculated as load per unit area.

$$p = F/A$$

Where,

p = Stress in N/mm^2
F = Applied Force in Newton
A = Cross-Sectional Area in mm^2

The material can undergo various types of stresses which are discussed next.

Tensile Stress

If the resistance offered by a body is against the increase in the size, the body is said to be under tensile stress.

Compressive Stress

If the resistance offered by a body is against the decrease in the size, the body is said to be under compressive stress. Compressive stress is just the reverse of tensile stress.

Shear Stress

The shear stress exists when two materials tend to slide across each other in any typical plane of shear on the application of force parallel to that plane. In other words, shear stress is generated in the body when force is applied parallel to the cross-section of the body.

Shear Stress = Shear resistance (R) / Shear area (A)

Strain

When a body is subjected to a load (force), its length changes. The ratio of change in the length to the original length of the member is called strain. If the body returns to its original shape on removing the load, the strain is called elastic strain. If the body remains distorted after removing the load, the strain is called plastic strain. The strain can be of three types, tensile, compressive, and shear strain.

Strain (e) = Change in Length (*dl*) / Original Length (*l*)

Elastic Limit

The maximum stress that can be applied to a material without producing the permanent deformation is known as the elastic limit of the material. If the stress is within the elastic limit, the material returns to its original shape and dimension on removing the external force. The following laws are used to define the response of elastic limit.

Hooke's Law

This law states that the stress is directly proportional to the strain within the elastic limit.

Stress / Strain = Constant (within the elastic limit)

Young's Modulus or Modulus of Elasticity

In case of axial loading, the ratio of intensity of the tensile or compressive stress to the corresponding strain is constant. This ratio is called Young's modulus, and is denoted by E.

E = p/e

Shear Modulus or Modulus of Rigidity

In case of shear loading, the ratio of shear stress to the corresponding shear strain is constant. This ratio is called Shear modulus, and it is denoted by C, N, or G.

Ultimate Strength

The maximum stress that a material withstands when subjected to an applied load is called its ultimate strength.

Yield Strength

The maximum stress that can be developed in a material without causing plastic deformation is called its yield strength.

Factor of Safety

The ratio of the ultimate strength to the estimated maximum stress in ordinary use (design stress) is known as factor of safety. It is necessary that the design stress is within the elastic limit, and to achieve this condition, the ultimate stress should be divided by a 'factor of safety'.

Lateral Strain

If a cylindrical rod is subjected to an axial tensile load, the length (l) of the rod will increase (dl) and the diameter ($\emptyset$) of the rod will decrease ($d\emptyset$). In short, the longitudinal stress will not only produce a strain in its own direction, but will also produce a lateral strain. The ratio dl/l is called the longitudinal strain or the linear strain, and the ratio $d\emptyset/\emptyset$ is called the lateral strain.

Poisson's Ratio

The ratio of the lateral strain to the longitudinal strain is constant within the elastic limit. This ratio is called the Poisson's ratio and is denoted by μ. The value of 'μ' lies between 0.0 to 0.5.

Poisson's ratio (μ) = Lateral Strain / Longitudinal Strain

Bulk Modulus

If a body is subjected to equal stresses along the three mutually perpendicular directions, the ratio of the direct stresses to the corresponding volumetric strain is found to be constant for a given material, when the deformation is within a certain limit. This ratio is called the bulk Modulus and is denoted by K.

Stress Concentration

The value of stress changes abruptly in the regions where the cross-section or profile of a structural member changes abruptly. The phenomenon of this abrupt change in stress is known as stress concentration and the region of the structural member affected by stress concentration is known as the region of stress concentration. The region of stress concentration needs to be meshed densely to get accurate results.

Bending

When a force is applied perpendicular to the longitudinal axis of a body, the body starts deforming. This phenomenon is known as bending. In case of bending, strains vary linearly from the centerline of the beam to the circumference. In case of pure bending, the value of strain is zero at the center line.

Bending Stress

When a non-axial force is applied on a structural member, some compressive and tensile stresses are developed in the member. These stresses are known as bending stresses.

Creep

At elevated temperature and constant load, many materials continue to deform, but at a slow rate. This behavior of materials is called creep. At a constant stress and temperature, the rate of creep is approximately constant for a long period of time. After a certain amount of deformation, the rate of creep increases, thereby causing fracture in the material. The rate of creep is highly dependent on both the stress and the temperature.

Degrees of Freedom (DOF)

The degrees of freedom is defined as the freedom allowed to a given object to move and rotate in any direction in space.

There are six DOFs for any object in 3-dimensional (3D) space: 3 translational DOFs (one each in the X,Y and Z directions) and 3 rotational DOFs (one rotation about each of the X,Y, and Z axes).

STRESS ANALYSIS USING Autodesk Inventor Nastran

To carry out Stress Analysis using Autodesk Inventor Nastran, you need to define the geometry on which you want to carry out the analysis. The Stress Analysis can be started by using the **Autodesk Inventor Nastran** tool available in the **Environments** tab.

A geometry or a model can be included in the analysis in two ways either by creating a new geometry or by opening an existing model in Inventor Professional. To create a new model, choose the **New** button from the **Launch** panel of the **Get Started** tab; the **Create New File** dialog box will be displayed. Select the desired template and create the model.

To open an existing part file, choose the **Open** button from the **Launch** panel of the **Get Started** tab; the **Open** dialog box will be displayed. Browse to the desired folder to open the file.

Now, as the geometry for the analysis is ready, you can invoke the Autodesk Nastran environment. To do so, choose the **Autodesk Inventor Nastran** tool from the **Environments** tab of the **Ribbon**; the **Autodesk Inventor Nastran** contextual tab will be displayed, as shown in Figure 20-3. Now you need to specify the initial settings for the analysis with the help of the **Default Settings** tool. This tool is discussed next.

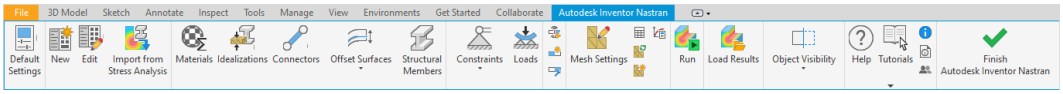


Figure 20-3 *The* ***Autodesk Inventor Nastran*** *contextual tab*

Defining Default Settings

Ribbon: Autodesk Nastran > System > Default Settings



To perform stress analysis, you first need to define the initial settings. To define the default settings, choose the **Default Settings** tool from the **System** panel of the **Autodesk Inventor Nastran** contextual tab; the **Default Settings** dialog box will be displayed, as shown in Figure 20-4. Alternatively, you can invoke this dialog by right-clicking in the Browser Bar and then choosing the **Default Settings** option from the shortcut menu. In this dialog box, you can specify a name for the analysis, the type of analysis to be carried out, design objectives, type of contacts to be used among components, and so on. The options in this dialog box are discussed next.

General

The options under this option are used to specify the settings for the **Nastran Wide Field Format**, **Nastran File Settings**, **Solution Settings**, **Database Settings**, **Mesher Settings** and **ADP Settings** options. You can use logging option for technical support of the software only. The Help option is used to switch between Online Help and the Local Help.

Display Options

The **Display Options** is used to display the constraints, loads, max/min on contour, meshes, coordinate systems, material orientation, concentrated mass, connectors, free edges, mesh control, and so on.

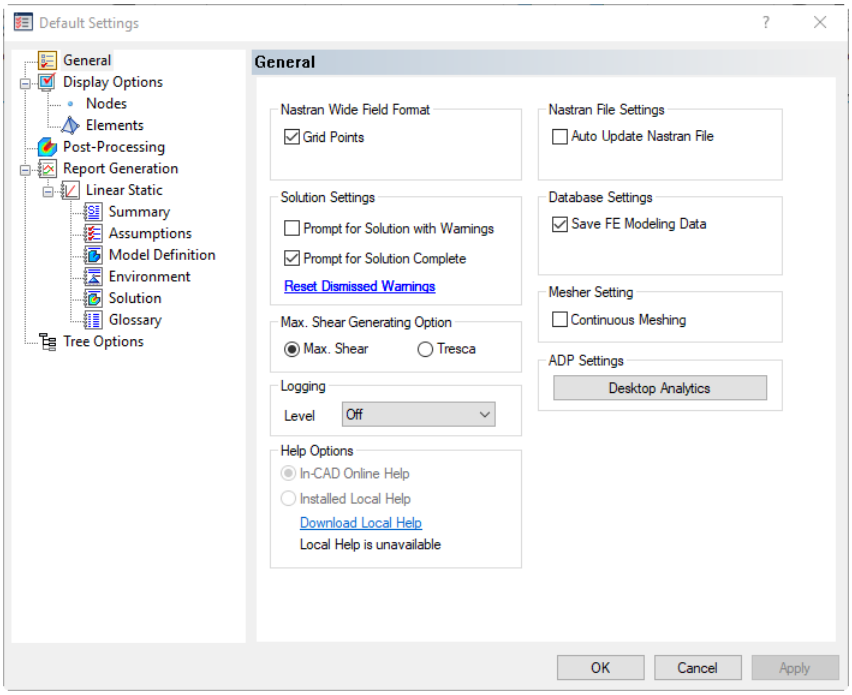


Figure 20-4 *The* ***Default Settings*** *dialog box*

Nodes

This sub node displays the nodes in the model. The **Display Nodes** and **Display Labels** check boxes are selected to display nodes with either the displacement coordinate system or id tag. You can also change the size and color with the help of the **Node Size** and **Node Color** options.

Elements

This sub node is used to display the labels for elements in the model by ID, material, or property.

Post-Processing

This node is used to define the default settings for post-processing options such as loading the result of the model, safety factor for the model, number of frames to be used in the animations, and so on.

Report Generation

The options in this node are used to define the settings for report generation of the preprocessor, solver, and postprocessor data of the problem after completing the solution. You can only generate the reports of Linear Static solutions.

Tree Options

This node is used to simplify the design tree by selecting and clearing the check boxes available in the **Tree Options** area. You can customize the tree options according to the requirement with the help of the **Defaults**, **Check All,** and **Uncheck All** options.

BROWSER BAR

The **Browser Bar** is the most important component of the Autodesk Inventor Nastran environment and is available below the **Ribbon** on the left of the drawing window. It displays all the operations performed during the analysis process in a sequence. All these operations are displayed in the form of a tree view. Figure 20-5 shows partial view of the default **Browser Bar**.

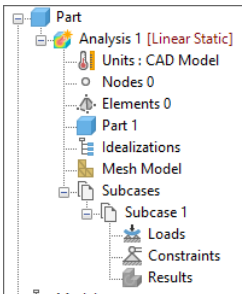


Figure 20-5 *Partial view of the default* ***Browser Bar***

NEW

This tool is used to define a new analysis. When you choose this tool, the **Analysis** dialog will be displayed, as shown in Figure 20-6. In this dialog box, you can specify a name of the analysis to be performed, the type of analysis to be performed, type of units for the analysis, type of contacts to be used in the component, model state of the component and so on. After you specify the required parameters, choose the **OK** button; the new analysis will be displayed in the Model tree.

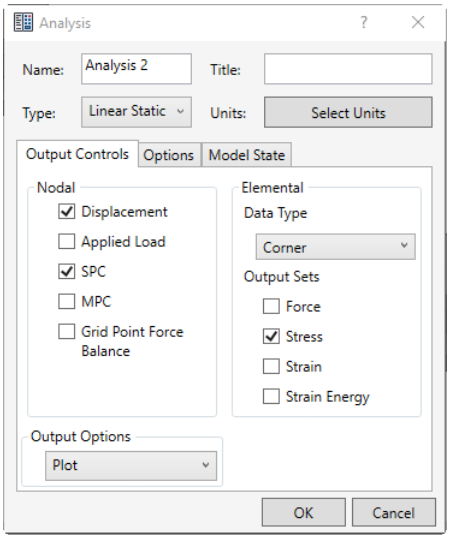


Figure 20-6 *The* ***Analysis*** *dialog box*

EDIT

This tool is used to modify the titles, types of the analysis, units, output control (Displacement, velocity, applied load, acceleration, and so on), model state of the components, contact type and so on of the current active analysis.

IMPORT FROM STRESS ANALYSIS

This tool is used to import the FEA model from Inventor Stress Analysis environment. Imported model includes loads, constraints, materials, midsurfaces, offset surfaces, mesh controls, surface contacts, and analysis settings.

MATERIAL PROPERTIES

In the analysis process, you need to assign material properties to the geometry created or imported. These properties are based on the analysis type and variation in material properties in 3D space. The material properties can be linear or non-linear.

Assigning Materials

Ribbon: Autodesk Inventor Nastran > Prepare > Materials



To assign materials to a model, choose the **Materials** tool available in the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Material** dialog box will be displayed, as shown in Figure 20-7. Alternatively, right-click on the **Materials** sub node under the **Model** node; a shortcut menu will be displayed. Choose the **New** option from the shortcut menu to invoke the **Material** dialog box.

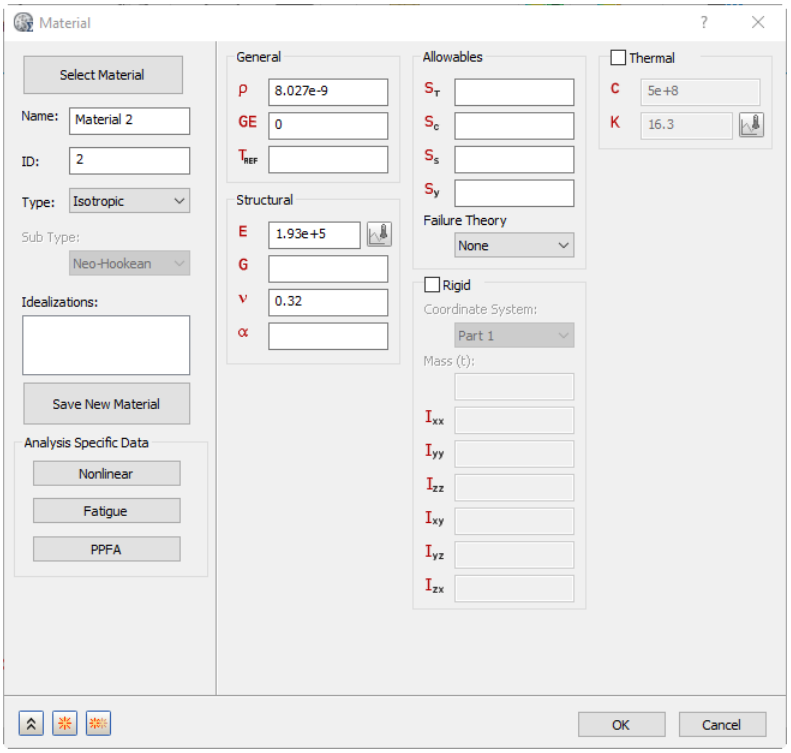


Figure 20-7 *The* ***Material*** *dialog box*

The options in the **Material** dialog box are discussed next.

Select Material

On selecting the **Select Material** option, the **Material DB** dialog box is displayed. You can also load the Autodesk Inventor material library. If you want to select another material database, then choose the **Load Database** button from the dialog box.

Name

The **Name** edit box is used to specify the name of the material of the model.

ID

As a new material is assigned to component, the ID of the material is automatically updated sequentially so that it is unique to each of the material entities.

Type

The **Type** drop-down list has nine material types that are supported in Autodesk Inventor Nastran. However, only Isotropic material data can be loaded from the material library. You can create and save your own material properties.

Sub Type
This drop-down list is used to specify material types such as, such as Neo-Hookean, Mooney-Rivlin, Yeoh, Ogden, and Polynomial. This drop-down list is activated only when **Hyperelastic** is selected from the **Type** drop-down list.

Save New Material
This button is chosen to create and save customised material property to filetype *.nasmat*.

Analysis Specific Data
This area is used to define the data related to specific analysis such as Nonlinear, Fatigue, PPFA.

General and Structural
The **General** and **Structural** areas display the properties of the selected material.

Allowables
This area is used to specify allowable values for tensile strength, compressive strength and shear strength, and Failure Theory.

After assigning the material type to the parts, choose the **OK** button from the **Material** dialog box to close it. The changes that are made to the model are displayed under the material node of the **Browser Bar**.

IDEALIZATION

Ribbon: Autodesk Inventor Nastran > Prepare > Idealization

Before performing further procedure, you need to idealize the model. Choose the **Idealization** tool available in the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Idealization** dialog box will be displayed, as shown in Figure 20-8.

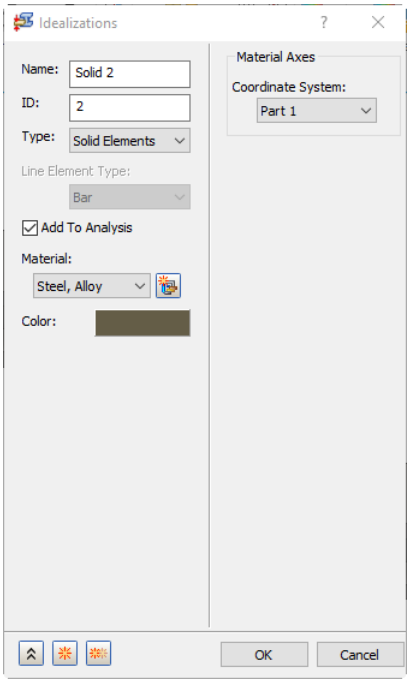


Figure 20-8 *The* ***Idealization*** *dialog box*

The various options in the **Idealization** dialog box are discussed next.

Name
The **Name** edit box is used to enter the name of the property assigned to a model.

ID
The **ID** display box is used to display the ID of the property which is automatically updated by Autodesk Inventor Nastran.

Type
The **Type** drop-down list is used to select the type of property for the components. The **Solid Elements** is selected for 3D linear and parabolic tetrahedral elements. The **Shell Elements** is selected for the 2D quadrilateral and triangular elements. The **Line Elements** is selected for 1D elements with 6 degrees of freedom.

Line Element Type Area
This area is activated only when **Line Element** is selected from the **Type** drop-down list. Three types of line elements are available namely Bar, Beam, Pipe.

Add to Analysis
The **Add to Analysis** check box is used to assign the idealization to the current analysis. This check box is activated by default.

New
The **New** button is selected to assign the new material for the component.

Color
This button is used to assign the color of the physical property which will be displayed as mesh color in the graphics window. After assigning the required parameters, choose the **OK** button from the **Idealization** dialog box to close it.

Connectors

Ribbon: Autodesk Inventor Nastran > Prepare > Connectors

Connectors are used to create connect between the selected components. To do so, invoke the **Connector** dialog box, choose the **Connectors** tool available in the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Connector** dialog box will be displayed, as shown in Figure 20-9. This dialog box is used to specify the name of the connector to be created. Also, you can select the type of connector created for the elements such as Rod, Cable, Spring, Rigid Body, and Bolt. After specifying the required parameters, choose the **OK** button from the **Connector** dialog box to close it.

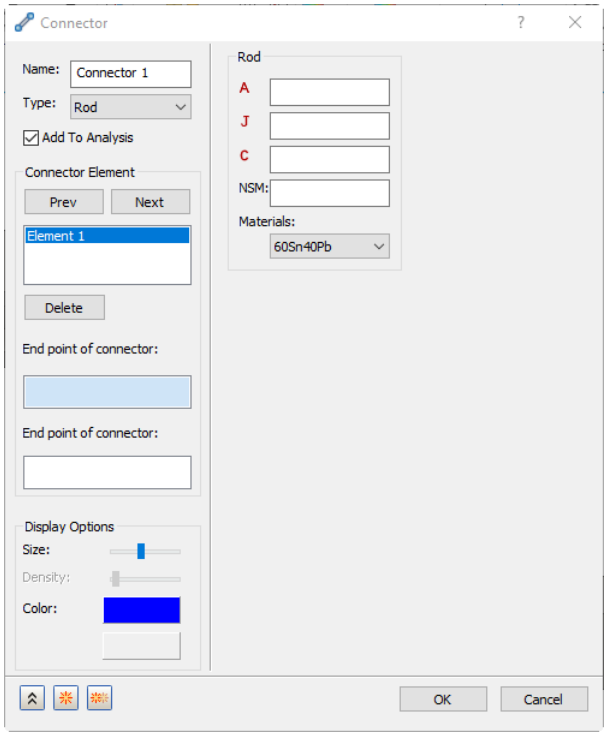


Figure 20-9 *The* ***Connectors*** *dialog box*

CONSTRAINTS

The constraints are applied to restrict the degrees of freedom of a component or group of components. Autodesk Inventor assumes the model under analysis to be fully constrained. This means there should not be any rigid body movements or free fall. The **Constraints** node in the **Browser Bar** shows the constraints applied to the system.

Applying Constraints

In Autodesk Inventor Stress Analysis, you can apply constraints by using the **Constraints** tool. To do so choose the **Constraints** tool available in the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Constraints** dialog box will be displayed, as shown in Figure 20-10. The options of this dialog box are discussed next.

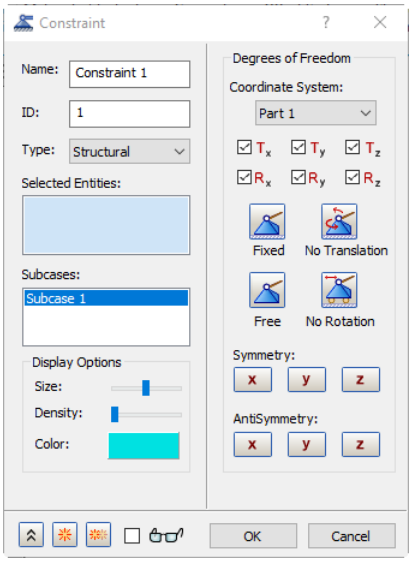


Figure 20-10 *The* ***Constraint*** *dialog box*

Name

This edit box is used to enter a description of the constraint. For example, left face is fixed.

ID

As a new constraint is assigned to component, the ID of a constraint is automatically updated sequentially so that it is unique to each of the constraints.

Type

This drop-down list has six constraint types such as Structural, Pin, Frictionless, Response Spectrum, Thermal, and Rigid (Explicit).

Selected Entities

This selection box is used to select the face, edge or vertex to specify the location of constarints. When you right-click on this selection box; a shortcut menu is displayed. You can choose the desired option from it.

Display Options
This area is used to edit the display option of the constraints such as size, density, and color.

Degree of Freedom
This area is used to specify the degrees of freedom of the nodes associated with the selected entities that are to be constrained.

Minimize/Maximize the dialog
This button is used to minimize or maximize the dialog box.

New
This button is used to add new constraint to the selected entities.

Create Duplicate
This button is used to create the duplicate of current constraint with same settings. You can apply the same type of constraint to different entities or apply a different type of constraint to the same entities. After setting the required parameters, choose the **OK** button from the **Constraint** dialog box to close it.

APPLYING LOADS

Ribbon: Autodesk Inventor Nastran > Setup > Loads

The **Loads** node in the **Browser Bar** displays all the loads that are applied to the model considered for the analysis. After the constraints are defined, it is very important that you apply the load for which you want to analyze the model. In Autodesk Inventor, only structural analysis is done and therefore only structural loads are available. The various loads that can be applied on the model are: Force, Pressure, Bearing load, Moment, Gravity, Remote Force, Distributed Load, Hydrostatic Load, and so on.

To apply load, you need to specify a point, an edge, or a face on which force needs to be applied. When applied on a vertex, the force load becomes point load, whereas when applied on an edge or a face, it becomes a uniformly distributed load. By applying a force load, you can simulate the behavior of the model under that particular load. To do so, choose the **Load** tool from the **Setup** panel of the **Autodesk Inventor Nastran** contextual tab, the **Load** dialog will be displayed, as shown in Figure 20-11. In this dialog box, you can specify the name of the Load, ID of the load, types of load, direction of the applied load and magnitude of the load.

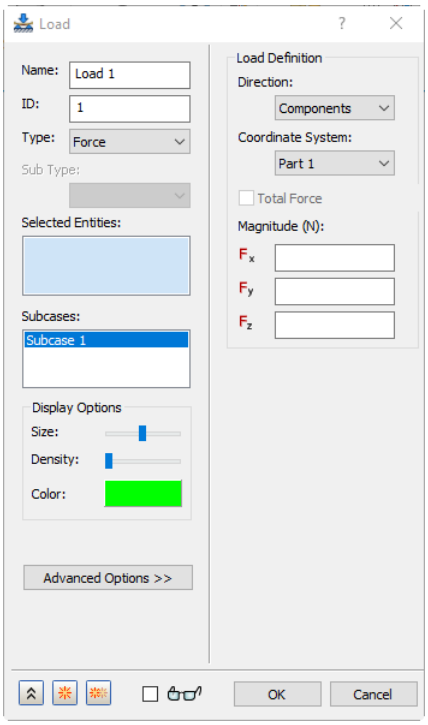


Figure 20-11 *The* ***Load*** *dialog box*

MESHING THE COMPONENT

A mesh is the discretization of a component into a number of small elements of defined size. In FEA, a geometry is divided into various small number of elements. As a result, a number of nodes are created which are used to connect the elements. A collection of these elements is called as mesh. The tools used for creating the mesh are discussed next.

Generate Mesh

Ribbon: Autodesk Inventor Nastran > Mesh > Generate Mesh



The **Generate Mesh** tool is used to create the default mesh in the component by using the default settings. These default settings are provided by the system based on the geometry to be meshed. To generate the default mesh, choose the **Generate Mesh** tool from the **Mesh** panel in the **Autodesk Inventor Nastran** contextual tab. You can also invoke this tool from the shortcut menu, which is displayed when you right-click on the **Mesh Model** node in the **Browser Bar**.

On choosing this tool, the **Mesh Progressing Bar** will be displayed, as shown in Figure 20-12 and mesh is generated on the component with default settings, refer to Figure 20-13.

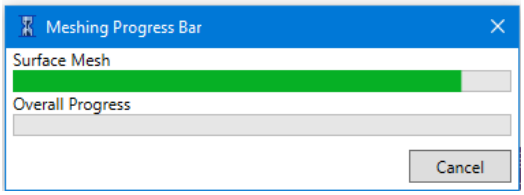


Figure 20-12 *The* ***Mesh*** *progress box*

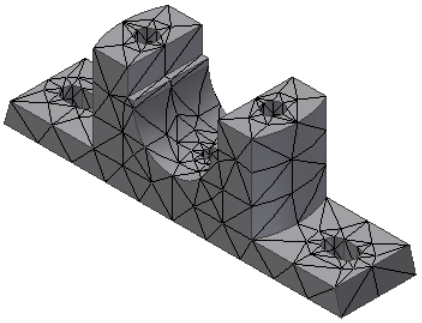


Figure 20-13 *Mesh generated with default settings*

Mesh Settings

Ribbon: Autodesk Inventor Nastran > Mesh > Mesh Settings

The **Mesh Settings** tool is used to set or edit the element length of the mesh. To do so, choose the **Mesh Settings** tool from the **Mesh** panel of the **Autodesk Inventor Nastran** contextual tab.

On choosing this tool, the **Mesh Settings** dialog box will be displayed, as shown in Figure 20-14. In this dialog box, you can set the average element size and minimum element size using the corresponding edit boxes.

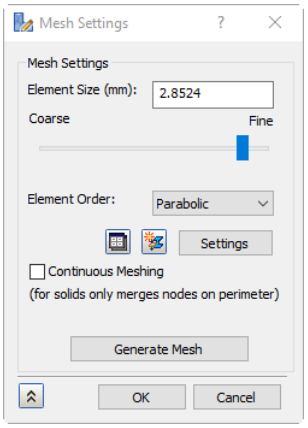


Figure 20-14 *The* ***Mesh Settings*** *dialog box*

SOLUTION PHASE OF ANALYSIS

Ribbon: Autodesk Inventor Nastran > Solve > Run

After meshing the model and applying boundary conditions to it, you need to solve the analysis problem for the applied boundary condition. In the solution phase, the time taken by the computer to perform calculations and solve the problem mainly depend upon the model size and the type of solution. To do so, choose the **Run** tool from the **Solve** panel of the **Autodesk Inventor Nastran** contextual tab; the output will be displayed in the Browser Bar. Also, the **Autodesk Inventor Nastran** dialog box showing the solution completion message will be displayed, as

shown in Figure 20-15. Choose the **OK** button from the message box; the Von-Mises stress result based on the specified settings will be displayed, as shown in Figure 20-16.

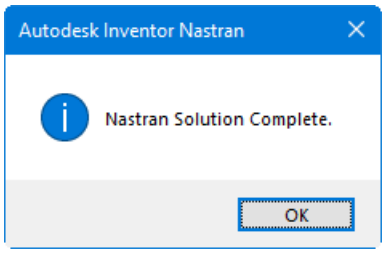


Figure 20-15 *The* ***Autodesk Inventor Nastran*** *message box*

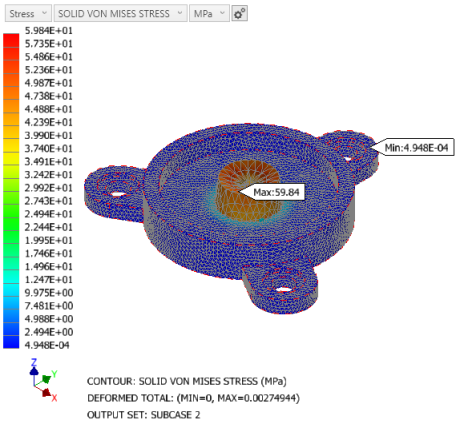


Figure 20-16 *The Von-Mises stress displayed*

TUTORIALS

Tutorial 1

In this tutorial, you will carry out linear static analysis on the model shown in Figure 20-17. The model for this tutorial can be downloaded from *www.cadcim.com*. The path of the file is as follows:

Textbook > CAD/CAM > Autodesk Inventor > Autodesk Inventor 2023 Professional for Designers > Input file > Model. *(Expected time: 45 min)*

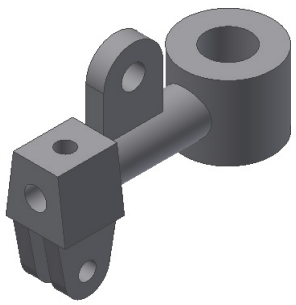


Figure 20-17 *The model for Tutorial 1*

The following steps are required to complete the tutorial.

a. Download the input file.
b. Define the settings.
c. Start a new study.
d. Apply constraints and loads.
e. Create mesh.
f. Generate the analysis report.
g. Save the report file.

Downloading the Assembly File

Before starting the tutorial, you need to download the model from the CADCIM website.

Download the c20_inv_2023_inp.zip file from www.cadcim.com. The path of the file is as follows: *Textbook > CAD/CAM > Autodesk Inventor > Autodesk Inventor 2023for Designers > Input file > Model.*

Defining Settings

After downloading the model, you need to open the model to define the initial settings.

1. Choose the **Autodesk Inventor Nastran** tool from the **Begin** panel of the **Environments** tab; a new environment with the **Autodesk Inventor Nastran** contextual tab is invoked.

2. Choose the **Default Settings** tool from the **System** panel of the **Autodesk Inventor Nastran** contextual tab; the **Define Settings** dialog box is displayed.

3. Retain the default settings and choose the **OK** button.

Starting a New Analysis

After defining the initial settings, the next step is to start the analysis process.

1. Choose the **New** tool from the **Analysis** panel of the **Autodesk Inventor Nastran** contextual tab; the **Analysis** dialog box is displayed.

2. By default, **Analysis X** is displayed in the **Name** edit box. Rename **Analysis X** to **Linear Static Analysis**.

3. Next type **Axial Loading** in the **Title** edit box.

4. Select the **Linear Static** option from the **Type** drop-down list if not selected by default.

5. Choose the **Select Units** button from the **Units** area; the **Units** dialog box is displayed. Select **CAD Model** from the **Units** dialog box. Choose the **OK** button from the **Units** dialog box to close it.

6. Retain the settings of rest of the parameters and choose the **OK** button; the **Linear Static Analysis** tree is displayed in the Browser Bar.

Applying the Material

Now, you need to apply the material to the model.

1. Choose the **Materials** tool from the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Material** dialog box is displayed.

2. Choose the **Select Material** button from the **Material** dialog box; the **Material DB** dialog box is displayed.

3. Choose **Load Database** button from the **Material DB** dialog box; the **Open** dialog box is displayed. Next, browse to C:\Program Files\Autodesk\Inventor Nastran 2023\In-CAD\ *Materials*, and select the *ADSK_materials.nasmat* file.

4. Choose the **Open** button from the **Open** dialog box and select **16-25-6 Stainless Steel** from the **Steel** node in the **Material DB** dialog box. Choose the **OK** button from this dialog box.

5. Enter **2.5e+008** for **Sy** and select **von Misses Stress** from the **Failure Theory** drop-down list.

6. Retain the default parameters and choose **OK** button from the **Material** dialog box.

Idealizations of the Model

Now, you need to idealize to the model.

1. Choose the **Idealizations** tool from the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Idealizations** dialog box is displayed.

2. Type **Solid Property** in the **Name** edit box and select the **Solid Elements** option from the **Type** drop-down list.

3. Make sure that **16-25-6 Stainless Steel** is selected in the **Material** drop-down list.

4. Retain the default parameters and choose the **OK** button from the dialog box; **Solid Property** is added under the **Idealization** node in the **Model** tree of the Browser Bar.

Applying Constraints

After applying material to the model, you need to apply constraint to one end of the model.

1. Choose the **Constraints** tool from the **Setup** panel of the **Autodesk Inventor Nastran** contextual tab; the **Constraint** dialog box is displayed.

2. Type **Fixed Left Face** in the **Name** edit box.

3. Select the **Structural** option from the **Type** drop-down list and choose the **Fixed** button from the **Degrees of Freedom** area.

4. Select the left face of the model and choose the **OK** button from the **Constraint** dialog box; the Fixed constraint is applied.

Applying the Load

After applying the material and constraints to the model, you need to apply load at the top right edge of the model.

1. Choose the **Loads** tool from the **Setup** panel of the **Autodesk Inventor Nastran** contextual tab; the **Load** dialog box is displayed.

2. Type **Axial Loading** in the **Name** edit box.

3. Next, choose the interior face of the left hole of the model.

4. Make sure the **Components** option is selected from the **Direction** drop-down list.

5. Enter **1000** in the **Fy** edit box in the **Magnitude(N)** area; the force arrow is modified according to the specified direction.

6. Choose the **OK** button from the **Load** dialog box to apply the load and to close it.

Creating and Refining the Mesh

After applying the constraints and loads to the model, you need to generate mesh for the model.

1. Choose the **Generate Mesh** tool from the **Mesh** panel of the **Autodesk Inventor Nastran** contextual tab; the **Meshing Progress Bar** is displayed. Once the meshing process is done, the meshed model is displayed in the drawing area, as shown in Figure 20-18.

   Notice that after generating the default mesh on the model, the quality of the mesh is not fine. To obtain more accurate results in the linear static analysis of the model, you need to generate a good quality mesh. The next step is to discretize the model in such a manner that a smooth and good quality mesh is obtained.

2. Next, choose the **Mesh Settings** tool from the **Mesh** panel of the **Autodesk Inventor Nastran** contextual tab; the **Mesh Settings** dialog box is displayed.

3. Enter **2** in the **Element Size(mm)** edit box. Next, choose the **Generate Mesh** button from the **Mesh Settings** dialog box; the **Meshing Progress Bar** is displayed. Once the meshing process is done, the new meshed model is displayed in the drawing area, as shown in Figure 20-19.

4. Choose the **OK** button from the dialog box.

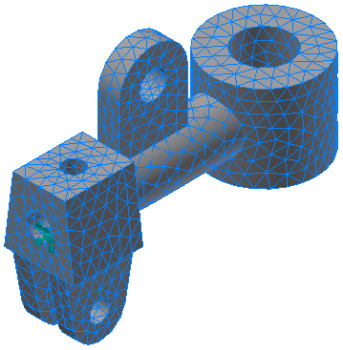


Figure 20-18 The meshed model with default mesh settings

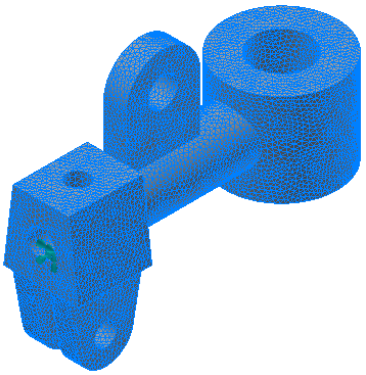


Figure 20-19 The meshed model with refined mesh settings

Solving the Linear Static Analysis

After applying all the required settings like constraints, loads, and mesh, you need to solve the analysis for the model.

1. Choose the **Run** tool from the **Solve** panel of the **Autodesk Inventor Nastran** contextual tab; the solution process starts and after the completion of this process the **Autodesk Inventor Nastran** dialog box is displayed.

2. Choose the **OK** button from the dialog box; the von Mises stress contour is displayed in the drawing area, as shown in Figure 20-20.

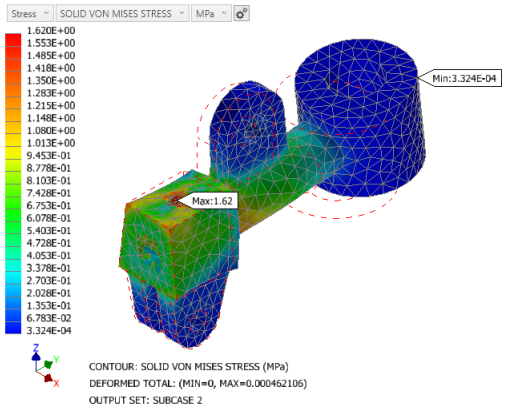


Figure 20-20 The resultant model with von Mises stress contour

Generating the Analysis Report

Now, you need to generate the complete report of the study for the model.

1. Choose the **Generate Report** option from the shortcut menu displayed on right-clicking on **Linear Static Analysis** in the Browser Bar; the **Report Wizard-Cover** dialog box is displayed.

2. Choose the **Next** button six times from the dialog box; the **Report Wizard-Glossary** dialog box.

3. Choose the **Finish** button from the dialog box; the **Save As** dialog box is displayed. Enter the name of the report file in the **File Name** edit box.

4. Specify the name of the file asTutorial1.iptand browse to the location *C :\Inventor_2023\c20.* Next, choose the **Save** button to save this model; the **Report** message box is displayed informing that report is successfully generated.

Tutorial 2

In this tutorial, you will carry out heat transfer and thermal stress analysis of a Flanged 90 Elbow, as shown in 20-21. The model for this tutorial can be downloaded from *www.cadcim.com*. The path of the file is as follows: *Textbook > CAD/CAM > Autodesk Inventor > Autodesk Inventor 2023 Professional for Designers > Input file > Model.*

(Expected time: 45 min)

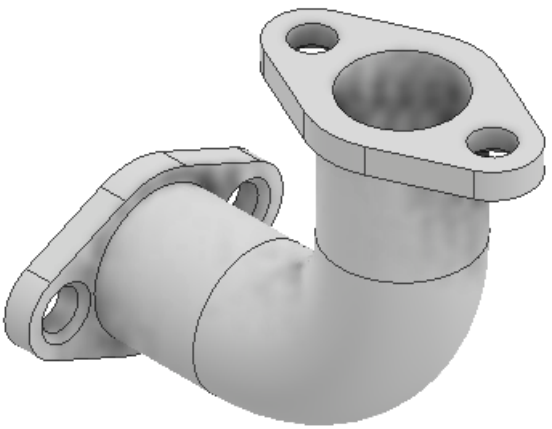


Figure 20-21 *The model for Tutorial 2*

The following steps are required to complete the tutorial.

a. Download the input file.
b. Define the settings.
c. Start a new study.
d. Apply material to the Flanged 90 Elbow
e. Idealization of the Flanged 90 Elbow
f. Create the mesh.
g. Solve the analysis.

Downloading the Assembly File

Before starting the tutorial, you need to download the model from the CADCIM website.

Download the file c20_inv_2023_inp.zipfile fromwww.cadcim.com. The path of the file is as follows: *Textbook > CAD/CAM > Autodesk Inventor > Autodesk Inventor 2023 for Designers > Input file>Model.*

Defining Settings

After creating the model, the next step is to define the initial settings.

1. Choose the **Autodesk Inventor Nastran** tool from the **Begin** panel of the **Environments** tab; a new environment with the **Autodesk Inventor Nastran** contextual tab is invoked.

2. Choose the **Default Settings** tool from the **System** panel of the **Autodesk Inventor Nastran** contextual tab; the **Define Settings** dialog box is displayed.

3. Retain the default settings and choose the **OK** button.

Starting a New Analysis

After defining the initial settings, the next step is to start the analysis procedure.

1. Choose the **New** tool from the **Analysis** panel of the **Autodesk Inventor Nastran** contextual tab; an **Analysis** dialog box is displayed.

2. By default, **Analysis X** is displayed in the **Name** edit box. Rename **Analysis X** to **Flanged 90 Elbow**.

3. Next type **NL SS Heat Transfer** in the **Title** edit box.

4. Select the **Non Linear Steady State Heat Transfer** option from the **Type** drop-down list.

5. Choose the **Select Units** button from the **Units** area; the **Units** dialog box is displayed. Select **CAD Model** from the **Units** dialog box. Choose the **OK** button from **Units** dialog box to close it.

6. Retain default settings of rest of the parameters and choose the **OK** button; the **Linear Static Analysis** tree is displayed in the Browser Bar.

Applying the Material

Now, you need to apply the material to the model.

1. Choose the **Materials** tool from the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Material** dialog box is displayed.

2. Choose the **Select Material** button from the **Material** dialog box; the **Material DB** dialog box is displayed.

3. Select **Load Database** button from the **Material DB** dialog box; the **Open** dialog box is displayed. Next, browse to *C:\Program Files\Autodesk\Inventor Nastran 2023\In-CAD\Materials,* and select the *ADSK_materials.nasmat file*.

4. Choose **Open** button from the **Open** dialog box and select **4130 Alloy Steel**, **tempered**, **Ultra High Strength from the Steel node in the Material DB dialog box. Choose the OK** button from this dialog box.

5. Next enter **150** in the **Tref** edit box under the **General** area.

6. Make sure that the **Thermal** check box is selected. Enter **40.33** in the **C** edit box and **0.00055555** in the **K** edit box.

7. Select the **None** option from the **Failure Theory** drop-down list.

8. Retain the default parameters and choose the **OK** button from the **Material** dialog box.

Idealizations of the model

Now, you need to apply idealizations to the model.

1. Choose the **Idealizations** tool from the **Prepare** panel of the **Autodesk Inventor Nastran** contextual tab; the **Idealizations** dialog box is displayed.

2. Type **Linear Solid Property** in the **Name** edit box and select the **Solid Elements** option from the **Type** drop-down list.

3. Make sure that **4130 Alloy Steel**, **tempered**, **Ultra High Strength** is selected in the **Material** drop-down list.

4. Retain the default parameters and choose the **OK** button from the dialog box; the **Linear Solid Property** is added under the **Idealization** node in the **Model** tree of the Browser Bar.

Applying the Load

After applying the material and constraints to the model, you need to apply load at the top right edge of the model.

1. Choose the **Loads** tool from the **Setup** panel of the **Autodesk Inventor Nastran** contextual tab; the **Load** dialog box is displayed.

2. Type **NLSSHT Loads** in the **Name** edit box.

3. Select **Heat Flux** from the **Type** drop-down list.

4. Next, select the inside surface of the flange; the surface gets highlighted in blue in color, as shown in Figure 20-22.

5. Enter **0.035** in the **Magnitude(mW/mm2)** in the **Load Definition** area.

6. Type **NLSST Loads** in the **Name** edit box.

7. Choose the **New** button from the dialog box and type **Convection** in the **Name** edit box.

8. Select **Convection** from the **Type** drop-down.

9. Select the outer surface as shown in Figure 20-23.

10. Select **Heat Flux** from the **Type** drop-down.

11. Enter **3.858E-5** in the **Convection Coefficient -h(mW/(mm2K))** edit box and **70 °F** in an **Ambient Temperature** edit box.

12. Again choose the **New** button from the dialog box to define the another load.

13. Type **Initial Temperature** in the **Name** edit box and select **Initial Condition** from the **Type** drop-down list.

14. Make sure that **Temperature** is selected in the **Sub Type** drop-down list.

15. Enter **70** in the **Temperature(K)** edit box in the **Load Definition** area.

16. Choose the **OK** button from the dialog box and save the model.

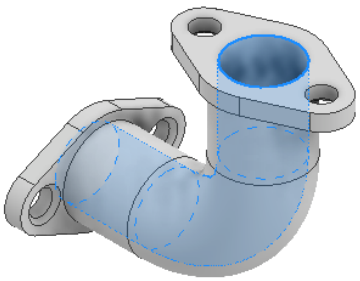


Figure 20-22 *The meshed model with default mesh settings*

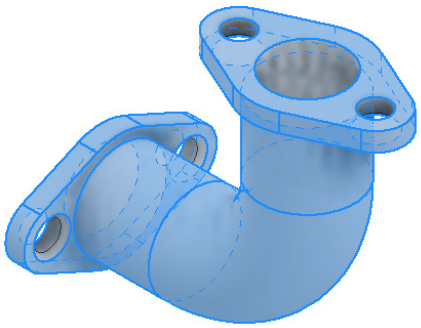


Figure 20-23 *The meshed model with refined mesh settings*

Creating and Refining the Mesh

After applying the constraints and loads to the model, you need to generate mesh for the model.

1. Choose the **Generate Mesh** tool from the **Mesh** panel of the **Autodesk Inventor Nastran** contextual tab; the **Meshing Progress Bar** is displayed. Once the meshing process is done, the meshed model is displayed in the drawing area, as shown in Figure 20-24.

Notice that after generating the default mesh on the model, the quality of the mesh is not fine. To obtain more accurate results in the linear static analysis of the model, you need to generate a good quality mesh. The next step is to discretize the model in such a manner that a smooth and good quality mesh is obtained.

2. Next, choose the **Mesh Settings** tool from the **Mesh** panel of the **Autodesk Inventor Nastran** contextual tab; the **Mesh Settings** dialog box is displayed.

3. Enter **2** in the **Element Size(mm)** edit box. Next, choose the **Generate Mesh** button from the **Mesh Settings** dialog box; the **Meshing Progress Bar** is displayed. Once the meshing process is done, the new meshed model is displayed in the drawing area, as shown in Figure 20-25.

4. Choose the **OK** button from the dialog box.

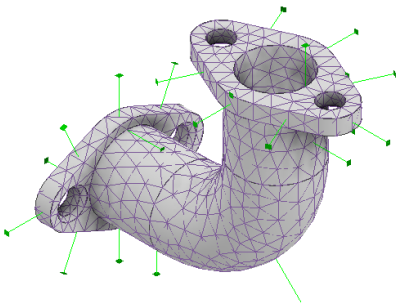


Figure 20-24 *The meshed model with default mesh settings*

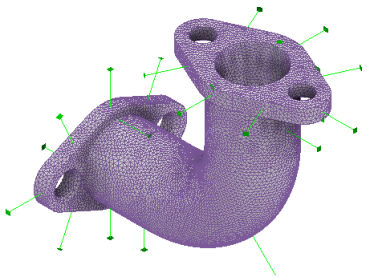


Figure 20-25 *The meshed model with refined mesh settings*

Solving the Analysis and Post Processing the Result

1. Right-click on the Flanged 90 Elbow in the Browser Bar; a shortcut menu is displayed.

2. Choose the **Solve in Nastran** option from the shortcut menu; the **Nastran File Progress Window** is displayed and solving process starts. After the completion of solution **Autodesk Inventor Nastran** dialog box is displayed.

3. Choose the **OK** button from the dialog box; the Von Mises stress contour is displayed in the drawing area, as shown in Figure 20-26.

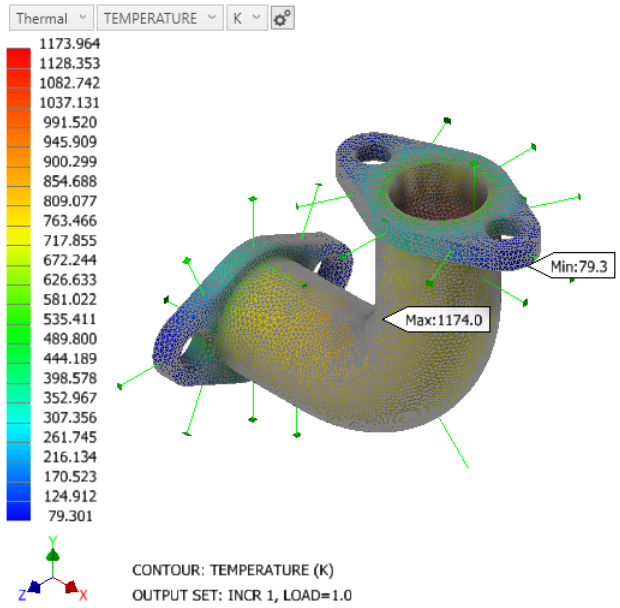


Figure 20-26 *The resultant model with von Mises stress contour*

Self-Evaluation Test

Answer the following the questions and then compare them to those given at the end of the chapter:

1. In Autodesk Inventor, you can start the Inventor Nastran environment by choosing the _____ tool.

2. After solving the simulation problems, the maximum stress value can be calculated by using the _____ tool.

3. The _____ tool is used to generate the solution of the analysis.

4. The _____ tool is used to set or edit the element length of the mesh.

5. The _____ button is used to create the duplicate of current constraint with same settings.

6. The _____ selection box is used to select the face, edge or vertex to specify the location of constraints.

Review Questions

Answer the following questions:

1. The _____ tool is used to generate mesh for a model.

2. FEA stands for _____.

3. The _____ tool is used to apply material to the model.

4. After solving the simulation, the minimum stress value can be located by using the _____ tool.

5. The tools in the _____ panel of the **Autodesk Inventor Nastran** contextual tab are used for generating and refining the mesh.

6. An automatically generated mesh can be refined by using the _____ tool.

EXERCISES

Exercise 1

In this exercise, you will perform the linear static analysis of the model. The holes of the model are fixed at its right end and load is applied at its left end, as shown in Figure 20-27. The dimensions are shown in Figure 20-28. The model is made up of alloy steel material.

(Expected time: 45 min)

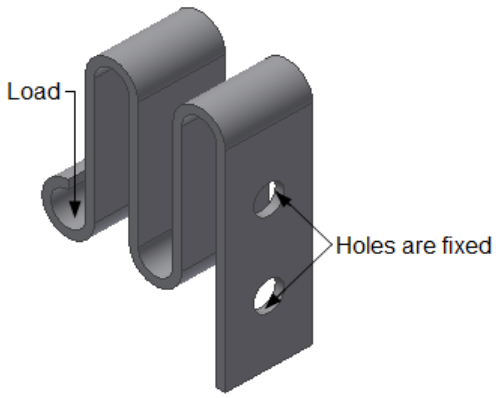


Figure 20-27 *The model for Exercise 1*

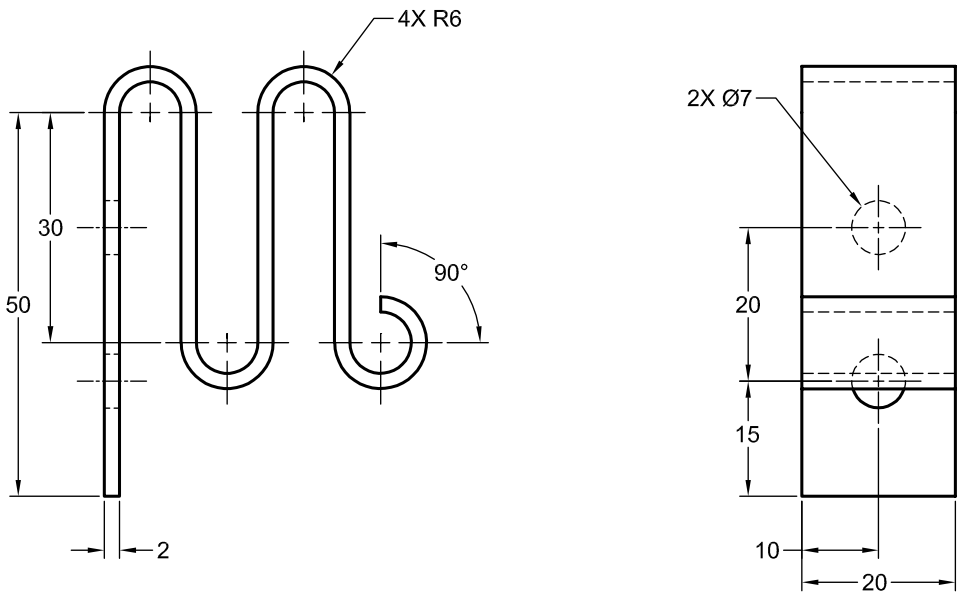


Figure 20-28 *The dimensions, boundary condition, and loads applied on the model*

Exercise 2

In this exercise, you will carry out heat transfer and thermal stress analysis of a modelas shown in 20-29. The dimensions applied to the model are shown in Figure 20-30. The model is made up of Steel Alloy material. **(Expected time: 45 min)**

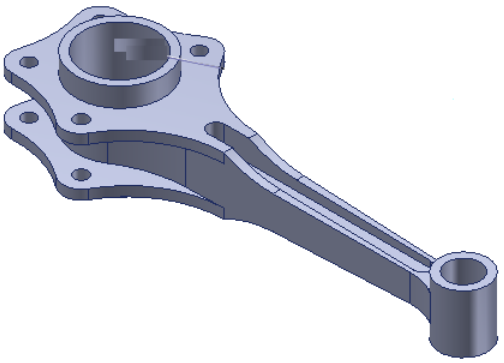


Figure 20-29 *The boundary conditions for the model*

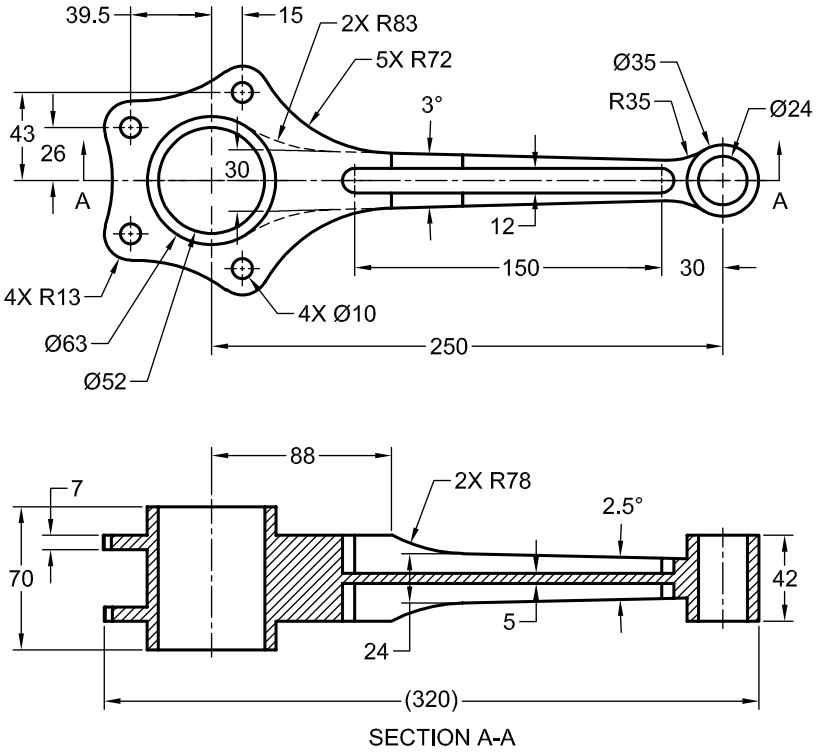


Figure 20-30 *The dimensions for the model*

Answers to Self-Evaluation Test

1. Autodesk Inventor Nastran, 2. Maximum Value, 3. Run, 4. Idealizations, 5. Create Duplicate 6. Selected Entities