

Autodesk Inventor Simulation 2010

Getting Started

The Autodesk logo is displayed in white text on a black rectangular background. The word "Autodesk" is written in a bold, sans-serif font, oriented vertically from bottom to top.

© 2009 Autodesk, Inc. All Rights Reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., in the USA and other countries: 3DEC (design/logo), 3December, 3December.com, 3ds Max, ADI, Alias, Alias (swirl design/logo), AliasStudio, AliasWavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, Autodesk Insight, Autodesk Intent, Autodesk Inventor, Autodesk Map, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backdraft, Built with ObjectARX (logo), Burn, Buzzsaw, CAiCE, Can You Imagine, Character Studio, Cinestream, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Create>what's>Next> (design/logo), Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, DesignStudio (design/logo), Design Web Format, Discreet, DWF, DWG, DWG (logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DXF, Ecotect, Exposure, Extending the Design Team, Face Robot, FBX, Filmbox, Fire, Flame, Flint, FMDesktop, Freewheel, Frost, GDX Driver, Gmax, Green Building Studio, Heads-up Design, Heidi, HumanIK, IDEA Server, i-drop, ImageModeler, iMOUT, Incinerator, Inferno, Inventor, Inventor LT, Kaydara, Kaydara (design/logo), Kynapse, Kynogon, LandXplorer, LocationLogic, Lustre, Matchmover, Maya, Mechanical Desktop, Moonbox, MotionBuilder, Movimento, Mudbox, NavisWorks, ObjectARX, ObjectDBX, Open Reality, Opticore, Opticore Opus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, ProMaterials, RasterDWG, Reactor, RealDWG, Real-time Roto, REALVIZ, Recognize, Render Queue, Retimer,Reveal, Revit, Showcase, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), SteeringWheels, Stitcher, Stone, StudioTools, Topobase, Toxik, TrustedDWG, ViewCube, Visual, Visual Construction, Visual Drainage, Visual Landscape, Visual Survey, Visual Toolbox, Visual LISP, Voice Reality, Volo, Vtour, Wire, Wiretap, WiretapCentral, XSI, and XSI (design/logo).

The following are registered trademarks or trademarks of Autodesk Canada Co. in the USA and/or Canada and other countries: Backburner,Multi-Master Editing, River, and Sparks.

The following are registered trademarks or trademarks of MoldflowCorp. in the USA and/or other countries: Moldflow, MPA, MPA (design/logo),Moldflow Plastics Advisers, MPI, MPI (design/logo), Moldflow Plastics Insight,MPX, MPX (design/logo), Moldflow Plastics Xpert.

All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

Published by:
Autodesk, Inc.
111 McInnis Parkway
San Rafael, CA 94903, USA

Contents

	Stress Analysis	1
Chapter 1	Get Started With Stress Analysis	3
	About Autodesk Inventor Simulation	3
	Learn Autodesk Inventor Simulation	4
	Use Help	4
	Use Stress Analysis Tools	5
	Understand the Value of Stress Analysis	6
	Understand How Stress Analysis Works	7
	Analysis Assumptions	7
	Interpret Results of Stress Analysis	9
	Equivalent or Von Mises Stress	9
	Maximum and Minimum Principal Stresses	10
	Deformation	10
	Safety Factor	10
	Frequency Modes	11
Chapter 2	Analyze Models	13
	Do a Static Stress Analysis	13
	Enter Environment and Create a Simulation	14
	Exclude Components	15
	Specify Material	16
	Add Constraints	16

	Add Loads	18
	Add Contact Conditions	19
	Generate a Mesh	20
	Run the Simulation	20
	Run Modal Analysis	21
Chapter 3	View Results	23
	Use Results Visualization	23
	Edit the Color Bar	25
	Read Stress Analysis Results	26
	Interpret Results Contours	26
	Animate Results	27
	Set Results Display Options	28
Chapter 4	Revise Models and Stress Analyses	31
	Change Model Geometry	31
	Change Solution Conditions	32
	Update Results of Stress Analysis	34
Chapter 5	Generate Reports	35
	Run Reports	35
	Interpret Reports	36
	Model Information	36
	Project Info	36
	Simulation	36
	Save and Distribute Reports	38
	Saved Reports	38
	Print Reports	38
	Distribute Reports	38
Chapter 6	Manage Stress Analysis Files	39
	Create and Use Analysis Files	39
	Understand File Relationships	39
	Resolve Missing Files	40
	Dynamic Simulation	41
Chapter 7	Get Started with Simulation	43
	About Autodesk Inventor Simulation	43
	Learning Autodesk Inventor Simulation	44
	Use Help	44
	Understand Simulation Tools	45

	Simulation Assumptions	45
	Interpret Simulation Results	45
	Relative Parameters	45
	Coherent Masses and Inertia	46
	Continuity of Laws	46
Chapter 8	Simulate Motion	47
	Understand Degrees of Freedom	47
	Understand Constraints	48
	Convert Assembly Constraints	49
	Run Simulations	52
Chapter 9	Construct Moving Assemblies	55
	Retain Degrees of Freedom	55
	Add Joints	57
	Impose Motion on Joints	58
	Run Simulations	59
Chapter 10	Construct Operating Conditions	61
	Complete the Assembly	61
	Add Friction	63
	Add a Sliding Joint	64
	Index	67

Stress Analysis

Part 1 of this manual presents the getting started information for Stress Analysis in the Autodesk Inventor® Simulation software. This add-on to the Autodesk Inventor assembly, part, and sheet metal environments provides the capability to analyze the static stress and natural frequency responses of mechanical designs.

Get Started With Stress Analysis

1

Autodesk Inventor® Simulation software provides a combination of industry-specific tools that extend the capabilities of Autodesk Inventor® for completing complex machinery and other product designs.

Stress Analysis in Autodesk Inventor Simulation is an add-on to the Autodesk Inventor assembly, part, and sheet metal environments.

Static Analysis provides the means to simulate stress, strain, and deformation.

Modal Analysis provides means to find natural frequencies of vibration and mode shapes of mechanical designs.

You can visualize the affects in 3D volume plots, create reports for any results, and perform parametric studies to refine your design.

This chapter provides basic information about the stress analysis environment and the workflow processes necessary to analyze loads and constraints placed on a part or assembly.

About Autodesk Inventor Simulation

Built on the Autodesk Inventor application, Autodesk Inventor Simulation includes several different modules. The first module included in this manual is Stress Analysis. It provides functionality for Structural Static and Modal analysis of mechanical product designs.

This manual provides basic conceptual information to help get you started and specific examples that introduce you to the capabilities of Stress and Modal Analysis in Autodesk Inventor Simulation.

Learn Autodesk Inventor Simulation

We assume that you have a working knowledge of the Autodesk Inventor Simulation interface and tools. If you do not, use Help for access to online documentation and tutorials, and complete the exercises in the Autodesk Inventor Simulation Getting Started manual.

At a minimum, we recommend that you understand how to:

- Use the assembly, part modeling, and sketch environments and browsers.
- Edit a component in place.
- Create, constrain, and manipulate work points and work features.
- Set color styles.

Be more productive with Autodesk® software. Get trained at an Autodesk Authorized Training Center (ATC®) with hands-on, instructor-led classes to help you get the most from your Autodesk products. Enhance your productivity with proven training from over 1,400 ATC sites in more than 75 countries. For more information about training centers, contact atc.program@autodesk.com or visit the online ATC locator at www.autodesk.com/atc.


We also recommend that you have a working knowledge of Microsoft® Windows® XP or Windows Vista®. It is desirable, but not required, to have a working knowledge of concepts for stress analysis of mechanical assembly designs.

Use Help

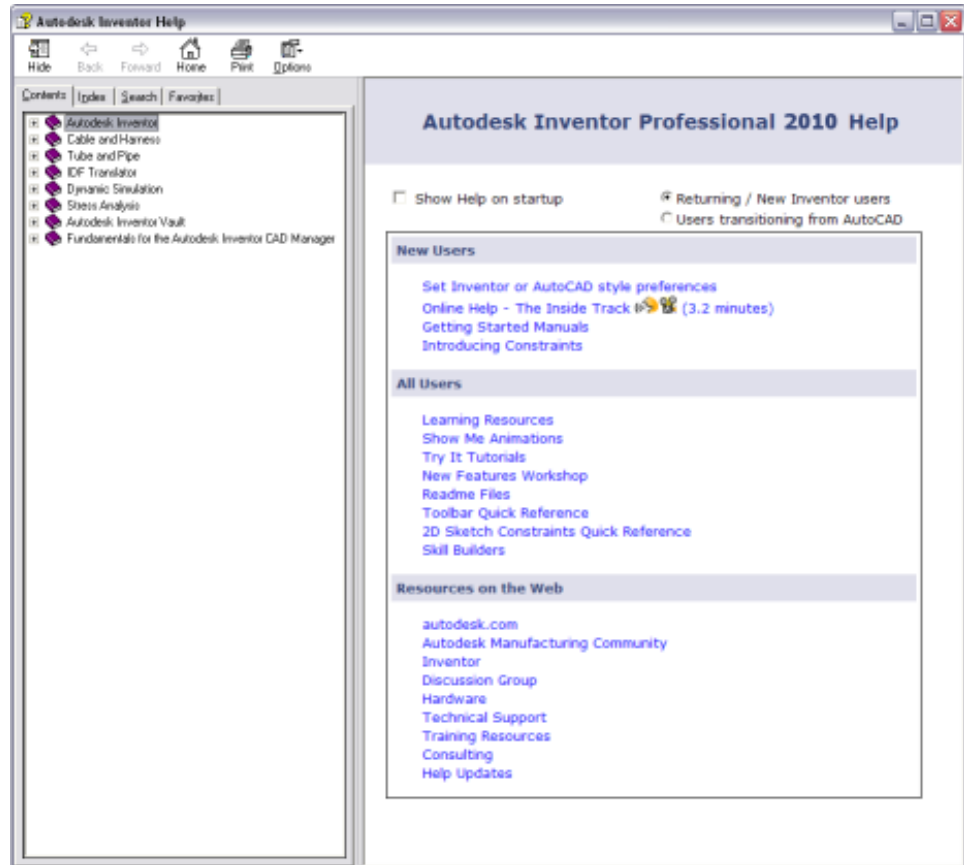
As you work, you may need additional information about the task you are performing. The Help system provides detailed concepts, procedures, and reference information about every feature in the Autodesk Inventor Simulation modules as well as the standard Autodesk Inventor Simulation features.

To access the Help system, use one of the following methods:

- Click Help ► Help Topics, and then use the Table of Contents to navigate to Stress Analysis topics.
- Press **F1** for Help with the active operation.

- In any dialog box, click  .

- In the graphics window, right-click, and then click **How To**. The **How To** topic for the current tool is displayed.



Use Stress Analysis Tools

Autodesk Inventor Simulation Stress Analysis provides tools to determine structural design performance directly on your Autodesk Inventor Simulation model. Autodesk Inventor Simulation Stress Analysis includes tools to place loads and constraints on a part or assembly and calculate the resulting stress, deformation, safety factor, and resonant frequency modes.

Enter the stress analysis environment in Autodesk Inventor Simulation with an active part or assembly.

With the stress analysis tools, you can:

- Perform a structural static or modal analysis of a part or assembly.
- Apply a force, pressure, bearing load, moment, or body load to vertices, faces, or edges of the model, or import a motion load from dynamic simulation.
- Apply fixed or non-zero displacement constraints to the model.
- Model various mechanical contact conditions between adjacent parts.
- Evaluate the impact of multiple parametric design changes.
- View the analysis results in terms of equivalent stress, minimum and maximum principal stresses, deformation, safety factor, or modal frequency.
- Add or suppress features such as gussets, fillets or ribs, re-evaluate the design, and update the solution.
- Animate the model through various stages of deformation, stress, safety factor, and frequencies.
- Generate a complete and automatic engineering design report in HTML format.

Understand the Value of Stress Analysis

Performing an analysis of a mechanical part or assembly in the design phase can help you bring a better product to market in less time. Autodesk Inventor Simulation Stress Analysis helps you:

- Determine if the part or assembly is strong enough to withstand expected loads or vibrations without breaking or deforming inappropriately.
- Gain valuable insight at an early stage when the cost of redesign is small.
- Determine if the part can be redesigned in a more cost-effective manner and still perform satisfactorily under expected use.

Stress analysis, for this discussion, is a tool to understand how a design performs under certain conditions. It might take a highly trained specialist a great deal of time performing what is often called a detailed analysis to obtain an exact answer with regard to reality. What is often as useful to help predict and improve a design is the trending and behavioral information obtained

from a basic or fundamental analysis. Performing this basic analysis early in the design phase can substantially improve the overall engineering process.

Here is an example of stress analysis use: When designing bracketry or single piece weldments, the deformation of your part may greatly affect the alignment of critical components causing forces that induce accelerated wear. When evaluating vibration effects, geometry plays a critical role in the natural frequency of a part or assembly. Avoiding, or in some cases targeting critical frequencies, can be the difference between failure and expected performance.

For any analysis, detailed or fundamental, it is vital to keep in mind the nature of approximations, study the results, and test the final design. Proper use of stress analysis greatly reduces the number of physical tests required. You can experiment on a wider variety of design options and improve the end product.

To learn more about the capabilities of Autodesk Inventor Simulation Stress Analysis, view the online demonstrations and tutorials.

Understand How Stress Analysis Works

Stress analysis is done using a mathematical representation of a physical system composed of:

- A part or assembly (model).
- Material properties.
- Applicable boundary conditions (loads, supports), contact conditions, and mesh, referred to as preprocessing.
- The solution of that mathematical representation (solving).
To find a result, the part is divided into smaller elements. The solver adds up the individual behaviors of each element to predict the behavior of the entire physical system by resolving a set of simultaneous algebraic equations.
- The study of the results of that solution is referred to as post-processing.

Analysis Assumptions

Extremely important to your simulation is the accuracy with which you model and specify the actual physical conditions (constraints, loads, materials, contact conditions). The accuracy of these conditions directly influences the quality of your results.

The stress analysis provided by Autodesk Inventor Simulation is appropriate only for linear material properties where the stress is directly proportional to the strain in the material (meaning no permanent yielding of the material). Linear behavior results when the slope of the material stress-strain curve in the elastic region (measured as the Modulus of Elasticity) is constant.

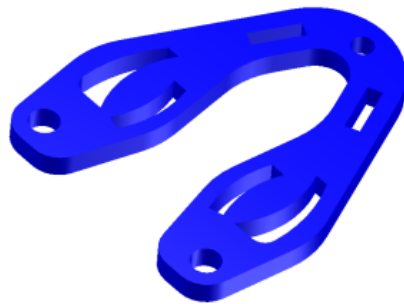
The total deformation is assumed to be small in comparison to the part thickness. For example, if studying the deflection of a beam, the calculated displacement must be less than the minimum cross-section of the beam.

The results are temperature-independent. The temperature is assumed not to affect the material properties.

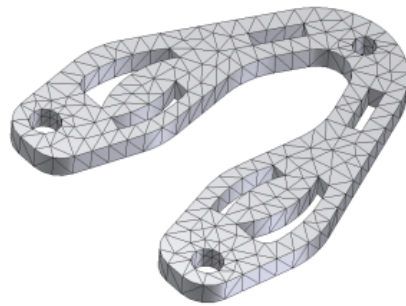
The following is a block (model) with well-defined mechanical and modal behaviors.



In this example of a simple part, the structural behavior would be difficult to predict solving equations by hand.



Here, the same part is broken into small elements, each with well-defined behaviors capable of being summed (solved) and easily interpreted (post-processed).



Interpret Results of Stress Analysis

The output of a mathematical solver is generally a substantial quantity of raw data. This quantity of raw data would normally be difficult and tedious to interpret without the data sorting and graphical representation traditionally referred to as post-processing. Post-processing is used to create graphical displays that show the distribution of stresses, deformations, and other aspects of the model. Interpretation of these post-processed results is the key to identifying:

- Areas of potential concern as in weak areas in a model.
- Areas of material waste as in areas of the model bearing little or no load.
- Valuable information about other model performance characteristics, such as vibration, that otherwise would not be known until a physical model is built and tested (prototyped).

The results interpretation phase is where the most critical thinking must take place. You compare the results (such as the numbers versus color contours, movements) with what is expected. You determine if the results make sense, and explain the results based on engineering principles. If the results are other than expected, evaluate the analysis conditions and determine what is causing the discrepancy.

Equivalent or Von Mises Stress

Three-dimensional stresses and strains build up in many directions. A common way to express these multidirectional stresses is to summarize them into an Equivalent stress, also known as the von-Mises stress. A three-dimensional solid has six stress components. If material properties are found experimentally

by an uniaxial stress test, then the real stress system is related by combining the six stress components to a single equivalent stress.

Maximum and Minimum Principal Stresses

According to elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. When the normal vector of a surface and the stress vector acting on that surface are collinear, the direction of the normal vector is called principal stress direction. The magnitude of the stress vector on the surface is called the principal stress value.

Deformation

Deformation is the amount of stretching that an object undergoes due to the loading. Use the deformation results to determine where and how much a part will bend, and how much force is required to make it bend a particular distance.

Safety Factor

All objects have a stress limit depending on the material used, which are presented as material yield or ultimate strengths. If steel has a yield limit of 40,000 psi, any stresses above this limit result in some form of permanent deformation. If a design is not supposed to deform permanently by going beyond yield (most cases), then the maximum allowable stress in this case is 40,000 psi.

A factor of safety can be calculated as the ratio of the maximum allowable stress to the equivalent stress (von-Mises), when using Yield Strength, and must be over 1 for the design to be acceptable. (Less than 1 means there is some permanent deformation.) When using Ultimate Strength, Maximum Principal stress is used to determine safety factor ratios.

Factor of safety results immediately points out areas of potential yield, where equivalent stress results always show red in the highest area of stress, regardless of how high or low the value. Since a factor of safety of 1 means the material is essentially at yield, most designers strive for a safety factor of between 2 to 4 based on the highest expected load scenario. Unless the maximum expected load is frequently repeated, the fact that some areas of the design go into yield does not always mean the part will fail. Repeated high load may result in a

fatigue failure, which is not simulated by Autodesk Inventor Simulation Stress Analysis. Always, use engineering principles to evaluate the situation.

Frequency Modes

Use modal frequency analysis to test a model for its natural resonant frequencies (for example, a rattling muffler during idle conditions, or other failures).

Each of these incidences may act on the natural frequency of the model, which, in turn, may cause resonance and subsequent failure. The mode shape is the displacement shape that the model adopts when it is excited at a resonant frequency. Autodesk Inventor Simulation calculates the natural frequencies of vibration and the corresponding mode shapes. It presents the mode shapes as results that you can view and animate. Dynamic response analysis is not offered at this time.

Analyze Models

2

After you define your model, you use the stress analysis environment to prepare the model for analysis. You define the materials, loads, and constraints for the condition you want to test, and establish contact conditions and mesh preferences. Then, you perform an analysis, also called simulation, of the model.

This chapter explains how to define materials, loads, constraints, contacts, and meshing, and then run your analysis.

Do a Static Stress Analysis

Use the stress analysis environment to analyze your assembly or part design and evaluate different options quickly. You can analyze a model under different conditions using various materials, loads and constraints (also called boundary conditions), and then view the results. You have a choice of performing a static analysis or a frequency (also called modal) analysis with associated mode shapes. After you view and evaluate the results, you can change your model and rerun the analysis to see the effect your changes produce.

Typical Stress Analysis workflow

- 1 Create Simulations and specify their properties.
- 2 Exclude components not required for simulation.
- 3 Assign materials. If you define a modal simulation, you can run it now. There is enough information to see the natural frequencies.
- 4 Add Constraints.
- 5 Add Loads.
- 6 Specify contact conditions, an optional step.
- 7 Specify and preview the mesh, an optional step.

- 8 Run the simulation.
- 9 View and Interpret the Results

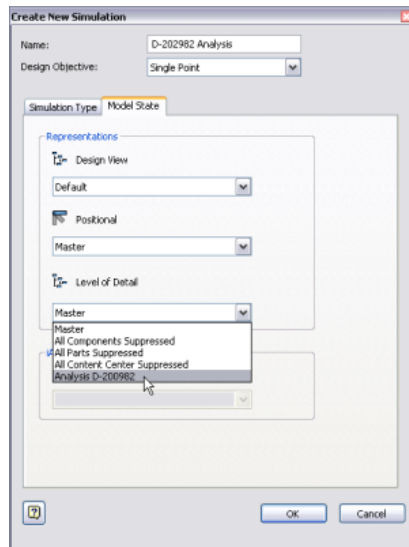
When you make modifications to the model or various inputs for the simulation, it can be necessary to update the mesh or other analysis parameters. A red lightning bolt icon next to the browser node indicates areas that need an update. Right-click the node and click Update to make them current with respect to the modifications. For the Results node, you must run the Simulate command to update results.

Enter Environment and Create a Simulation

You enter the Stress Analysis environment from the assembly, part, or sheet metal environments.

To enter the environment and create a new simulation:

- 1 Open the model you want to analyze. By default you are in the modeling environment.
- 2 On the ribbon, click Environments tab ► Begin panel ► Stress Analysis. The Stress Analysis tab displays.
- 3 On the ribbon, in the Manage panel ► Create Simulation.
You can create multiple simulations within the same document. Each simulation can use different materials, constraints, and loads.
- 4 Specify the simulation properties. Specify a name, type of simulation, and on the Model State tab, the model representation to use for the simulation.



5 Click OK. The new simulation populates the browser with analysis nodes.

Exclude Components



In assemblies, some components have no bearing on the simulation.

You can exclude the components. Right-click the component node and click Exclude from simulation.

Exclusion in a simulation has no effect on the assembly in the modeling environment.

In parts, you can exclude part features, such as small fillets and cosmetic features that have no bearing on the performance of the part.

Specify Material

The stress analysis environment provides the means to override materials for any component. The default material provided in Inventor templates is not completely defined for simulation purposes. When modeling your components, use materials that are appropriate and completely defined, particularly if you are going to use simulation.



- 1 Click Assign Materials. This step is optional based on the materials used for the components. If all materials are completely defined materials, you can forego material overrides.
- 2 In the dialog box, specify an override material for the components as needed. The override material is listed in the third column. Use the pulldown list to see what materials are available.
- 3 Make all necessary material overrides and failure criteria choices (Yield or Ultimate Strength) used for Safety Factor calculations, then click OK.


Inventor materials are maintained through the Styles and Standards editor. You can modify existing materials or define new ones according to your need. You can access the editor from the Assign Materials dialog box or by clicking Manage tab ► Styles and Standards panel ► Styles Editor.

Add Constraints

You add constraints to mimic environmental conditions. Constraint instances are child nodes of the browser Constraints node. Double-click a constraint node to edit the constraint.

NOTE Constraints are a crucial part of building a simulation model and can greatly affect the end results of the simulation. Take time to consider them carefully and accurately represent the physical conditions,

	Constraint	Constraint-Specific Information
	Fixed Constraint	Apply a fixed constraint to a face, edge, or vertex in the part. Apply a fixed constraint to establish zero or non-zero displacement on a part.
	Pin Constraint	Apply a pin constraint on cylindrical faces. Apply pin constraints to prevent cylindrical faces from moving or deforming in combinations of radial, axial, or tangential directions.

Constraint	Constraint-Specific Information
	Frictionless Constraint Apply a frictionless constraint to a flat or cylindrical surface in the part. Frictionless constraints prevent the surface from moving or deforming in the normal direction relative to the surface.

To add a constraint:

- 1 Click the constraint command corresponding with the type of constraint you want to assign.
- 2 The select command is active and you can begin selecting the geometry related to the constraint type. You can expand the dialog box to access advanced settings.

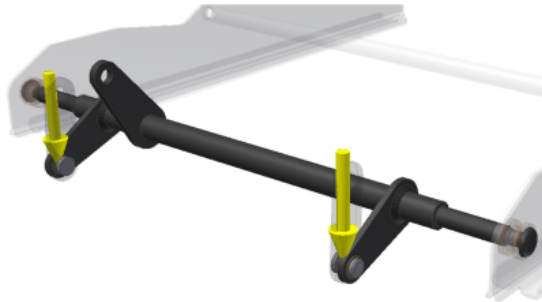
If you right-click a constraint in the browser, you can:

- Edit the constraint. The appropriate dialog box displays so that you can make changes.
- View reaction forces. Values are zero until a simulation is run.
- Suppress the constraint.
- Copy and Paste between simulations within the same document.
- Delete the constraint.






To rename an item in the browser, click it, pause, click it a second time, enter a new name, and then press ENTER.


NOTE For some types of simulations you define, constraints are not required.

Add Loads



To simulate conditions your design can encounter, you add force loads to areas where such forces can be encountered. There are a variety of load types to use. The following list explains the available load types.

Load	Load-Specific Information
 Force	Apply a force to a set of faces, edges, or vertices. When the force location is a face, the direction is automatically set to the normal of the face, with the force pointing to the inside of the part. Define the direction planar faces, straight edges, and axes.
 Pressure	Pressure is uniform and acts normal to the surface at all locations on the surface. Apply pressure only to faces.
 Bearing Load	Apply a bearing load only to cylindrical faces. By default, the applied load is along the axis of the cylinder and the direction of the load is radial.
 Moment	Apply a moment only to faces. Define direction using planar faces, straight edges, two vertices, and axes.
 Body Loads	Specifies the linear acceleration for the model using a face as input. Cylindrical selections provide an axial direction. You can only apply one body load per analysis.

Load	Load-Specific Information
	Gravity Specifies the direction of gravitational load on the model. Select a face to define the direction or use Vector Components to precisely control the direction. Cylindrical selections provide an axial direction.

To add a load, you must:

- 1 Click the load command corresponding to the load type you want to add.
- 2 The selection command is active so you can select the geometry appropriate to the load you are defining.
- 3 Specify the load parameters. When needed, expand the dialog box to access the advanced settings.

Double-click the load node in the browser to modify it. Alternatively, you can right-click the load node and click Edit [type] constraint.

Add Contact Conditions

In assemblies, various contact conditions may exist. These are automatically detected when using the Automatic Contacts command. The simulation properties specify the tolerance and type of contact that will be automatically assigned.

Review the contacts that are generated to ensure that these accurately represent the physical interactions of your model. Only one contact type can serve as the default for automatically inferred contacts, so some modification afterward may be necessary.

Automatic Contacts

To automatically add contact conditions, click the Automatic Contacts command. Alternatively, right-click the Contact node and click Automatic Contacts.

Manual Contacts

At times it is necessary to manually add contacts. To manually add contact conditions:

- 1 On the ribbon, click Stress Analysis tab ► Contacts panel ► Manual.

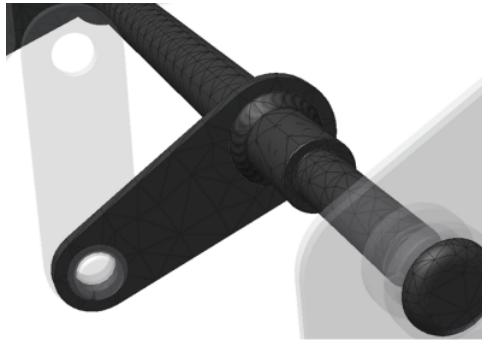
- 2 Specify the contact type.
- 3 Select the appropriate entities for the contact type. If other components are obscuring the component you want to select use Part selection option to select the part first, then refine your selection thereafter.

Generate a Mesh

You can accept the default mesh settings and proceed right to the simulation. At times there will be areas where you would like a mesh with greater density. To manage this you can adjust the mesh settings or use a local mesh control.

If you want to view the mesh settings, click the Mesh Settings command in the Prepare panel. You can specify the mesh settings you want for the simulation.

After you define the meshes, click Mesh View to produce the mesh. The mesh is generated as an overlay atop the model geometry.



Local Mesh Control

To apply a local mesh control, click the Local Mesh Control command in the Prepare panel. Then, select the face where the mesh will be applied and specify the mesh settings for the local control.

Run the Simulation

After you define the parameters for the analysis, you can run the simulation. On the ribbon, click Stress Analysis tab ► Solve panel ► Simulate.

In the Simulate dialog box, you can expand the More section to see if there are any process related notifications or warnings.

When ready, click Run to start the simulation calculations.

Run Modal Analysis

In addition to the stress analysis, you can perform a modal frequency analysis to find the natural frequencies at which your part vibrates, and the mode shapes at those frequencies. Like stress analysis, modal analysis is available in the stress analysis environment.

You can do a natural frequency analysis independent of a stress analysis. You can do a frequency analysis on a prestressed structure, in which case you can define loads on the part before the analysis. You can also find the natural frequencies of an unconstrained model.

Workflow: Run a modal analysis

- 1 Enter the stress analysis environment.
- 2 Start a new simulation, specifying Modal Analysis as the simulation type.
- 3 Verify that the material used for the part is suitable, or override the unsuitable with appropriate materials.
- 4 Apply the necessary constraints (optional).
- 5 Apply any loads (optional).
- 6 Make adjustments to the mesh settings and preview the mesh (optional).
- 7 Click Simulate and in the dialog box, click Run.

The results for the first eight frequency modes are inserted under the Results folder in the browser. For an unconstrained part, the first six frequencies are essentially zero.

- 8 To change the number of frequencies displayed right click the Simulation node (near the top of the browser), and select Edit Simulation Properties. In the dialog box specify the number of modes to find.

After you complete all the required steps, the Update notification is displayed in the browser beside those sections that need updates. Right-click the node and click Update. On the Results node, right-click the node and click Simulate.

View Results

3

After you analyze your model under the stress analysis conditions that you defined, you can visually observe the results of the solution.

This chapter describes how to interpret the visual results of your stress analyses.

Use Results Visualization

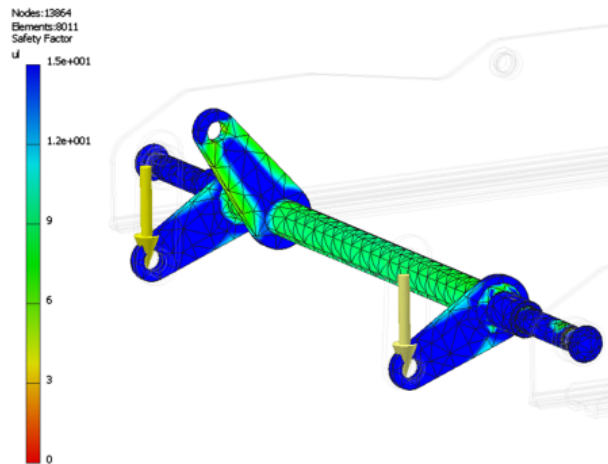
When the simulation completes its computations, the graphics region updates to show:

- 3D Volume plot and result type.
- Smooth Shading showing the distribution of stresses.
- Color bar indicating the stress range.
- Mesh Information including the number of nodes and elements.
- Unit information.
- Result browser node is populated with child nodes for the various results based on the analysis type.

For Static Analysis, the default result is Von Mises Stress and for Modal Analysis, the default is Frequency 1. View the results through the use of the display commands and the Results nodes in the browser. These tools help you visualize the magnitude of the stresses that occur throughout the component, the deformation of the component, and the stress safety factor. For modal analysis, you visualize the natural frequency modes.

Display commands are enabled and located in the Display panel of the Stress Analysis tab. The default display mode smooths contours.

The various results sets are seen by expanding the Result node to reveal the child nodes. For example, when you run a static analysis, child result nodes for Von Mises Stress, 1st principal stress, Displacement, Safety Factor, and so on populate the browser.



To view the different results sets, double-click the browser node. While viewing the results, you can:

- Change the color bar to emphasize the stress levels that are of concern.
- Compare the results to the undeformed geometry.
- View the mesh used for the solution.
- Use Contour Shading.
- Display the Maximum and Minimum Results locations.
- Apply the same scale across results sets. This becomes relevant when hiding various parts from the result view or when dealing with multiple configurations in parametric studies.
- Display the boundary conditions.
- Change the displacement scale to exaggerate the displacement.
- Animate the displacement through a series of steps.
- Create a video of the displacement animation.
- View 2D Convergence Plots (result accuracy curve).

- Probe for values at specific points.

Edit the Color Bar

The color bar shows you how the contour colors correspond to the stress values or displacements calculated in the solution. You can edit the color bar to set up the color contours so that the stress/displacement is displayed in a way that is meaningful to you.

Edit the color bar


- 1 On the ribbon, click Stress Analysis tab ► Display panel ► Color Bar. By default, the maximum and minimum values shown on the color bar are the maximum and minimum result values from the solution. You can edit the maximum and minimum values to adjust the way the bands appear.


- 2 To edit the maximum and minimum critical threshold values, clear the check box next to the item you want to modify. Edit the values in the text box. Click OK to complete the change.

To restore the default maximum and minimum critical threshold values, check the corresponding box next to the item.

The levels are initially divided into seven equivalent sections, with default colors assigned to each section. You can select the number of contour colors in the range of 2 to 12.

When using smooth shading, only 5 colors are used and these controls are disabled.

- 3  To increase or decrease the number of colors, click Increase Colors and Decrease Colors. You can also enter the number of colors you want in the text box.

- 4  You can view the result contours in different colors or in shades of gray. To view result contours on the grayscale, click Grayscale under Color Type.

NOTE It does not work for safety factor.

- 5 By default, the color bar is positioned in the upper-left corner. Select an appropriate option under Position to place the color bar at a different location.
- 6 For Size, select an appropriate option to resize the color bar, and then click OK.

The color bar settings are applied on a per result basis.

Read Stress Analysis Results

When the analysis is complete, you see the results of your solution. If you did a stress analysis, the Von Mises Stress results set displays. If your initial analysis is a natural frequency analysis, the results set for the first mode displays. To view a different results set, double-click that results set in the browser pane. The currently viewed results set has a check mark next to it in the browser. You always see the undeformed wireframe of the part when you are viewing results.

Interpret Results Contours

The contour colors display in the results corresponds to the value ranges shown in the legend. In most cases, results displayed in red are of most interest, either because of their representation of high stress or high deformation, or a low factor of safety. Each results set gives you different information about the effect of the load on your part.

Von Mises Stress

Von Mises stress results use color contours to show you the stresses calculated during the solution for your model. The deformed model is displayed. The color contours correspond to the values defined by the color bar.

1st Principal Stress

The 1st principal stress gives you the value of stress that is normal to the plane in which the shear stress is zero. The 1st principal stress helps you understand the maximum tensile stress induced in the part due to the loading conditions.

3rd Principal Stress

The 3rd principal stress acts normal to the plane in which shear stress is zero. It helps you understand the maximum compressive stress induced in the part due to the loading conditions.

Displacement

The Displacement results show you the deformed shape of your model after the solution. The color contours show you the magnitude of deformation from the original shape. The color contours correspond to the values defined by the color bar.

Safety Factor

Safety factor shows you the areas of the model that are likely to fail under load. The color contours correspond to the values defined by the color bar.

Frequency Modes








You can view the mode plots for the number of natural frequencies that you specified in the solution. The modal results appear under the Results node in the browser. When you double-click a frequency mode, the mode shape displays. The color contours show you the magnitude of deformation from the original shape. These are modal deformations and their magnitude is relative and cannot be taken as the actual deformation. The frequency of the mode shows in the legend. It is also available as a parameter.






Animate Results

Use Animate Displacement to visualize the part through various stages of deformation. You can also animate stress, safety factor, and deformation under frequencies.

Set Results Display Options

While viewing your results, you can use the following commands located on the Result and Display panels to modify the features of the results display for your model.

	Command	Used to
	Same Scale	Maintains the same scale while viewing different results.
	Color Bar	Displays the Color Bar settings dialog box where you adjust the color bar display parameters.
	Smooth Shading	Displays color changes using a blended transition.
	Contour Shading	Displays color changes using a strict banding between colors.
	No Shading	Turns off the Shaded Results display.
	Maximum	Turns on and off the display of the point of maximum result in the mode.
	Minimum	Turns on and off the display of the point of minimum result in the model.
	Boundary Condition	Turns on and off the display of the load symbols on the part.

Command	Used to
	<p>Probe</p> <p>Activates the Probe command. You place probes as needed in areas of interest to display the stress values for that point.</p>
	<p>Display Probe Labels</p> <p>Toggles the visibility of probe labels.</p>
	<p>Displacement Scale</p> <p>Displays a preset list of displacement exaggeration scales. Choose a scale that suits your need.</p>
	<p>Mesh View</p> <p>Displays the element mesh used in the solution in conjunction with the result contours. Also displays the mesh over the undeformed model.</p>
	<p>Animate Displacement</p> <p>Animates the displacement for the current result type, displacement scale, and the progressive results of the stress.</p>

Use the Displacement Scale pull-down list to adjust the deformed shape exaggeration. Select Actual to see the deformation to scale. Since the deformations are often small, the various automatic options exaggerate the scale so that the shape of the deformation is more pronounced.

Use the Display commands to set the shaded style to contour, smooth, or no shading. The legend shows while contours are off.

The values of all of the display options for each results set are saved for that results set.

Revise Models and Stress Analyses

4

After you run a simulation for your model, you can evaluate how changes to the model or analysis conditions affect the results of the simulation.

This chapter explains how to change simulation conditions for the model and rerun the simulation.

Change Model Geometry

After you run an analysis on your model, you can change the design of your model. Rerun the analysis to see the effects of the changes.

Edit a design and rerun analysis

- 1 In the browser, right-click the part or assembly you want to edit and click Open.
This opens the component in another window where you can make changes. At the bottom of the window near the status bar there is a tab for each open document. For the purposes of this discussion, we talk about a part edit.
- 2 In the browser, expand the feature node that you want to edit.
- 3 In the browser, right-click the sketch for the feature that you want to edit and click Show Dimensions. The dimensions for that feature display over the model.
- 4 Double-click the dimension that you want to change, enter the new value in the text box, and then click the green check mark. The sketch updates.
- 5 In the Quick Access Toolbar (QAT) click the Update model command.

- 6 At the bottom of the window, click the assembly tab. Your component is updated.
- 7 Some portions of the simulation may now be out of date with reference to the change. You must update these in order to have current analysis data. If an update is necessary, right-click the Contacts node, and click Update.
- 8 Repeat step 7 for each area that requires it. Then click Simulate to update the results.

After you update the simulation, the load glyphs relocate if the feature that they were associated with moved as a result of the geometry change. The direction of the load does not change, even if the feature associated with the load changes orientation.

Change Solution Conditions

After you run an analysis on your model, you can change the conditions under which the results were obtained. Rerun the analysis to see effects of the changes. You can edit the loads and constraints you defined, add new loads and constraints, or delete loads and constraints. To change your simulation conditions, enter the stress analysis environment if you are not already in it.

Delete a load or constraint

- In the browser, right-click a load or constraint, and then select Delete from the menu.

Add a load or constraint


- On the Stress Analysis tab, select the command and follow the same procedure you used to create your initial loads and constraints.

Edit a load or constraint

- 1 In the browser, right-click a load or constraint, and then select Edit from the menu.
The same dialog box you used to create the load or constraint displays. The values on the dialog box are the current values for that load or constraint.

- 2 Click the selection arrow on the left side of the dialog box to enable feature picking.
You are initially limited to selecting the same type of feature (face, edge, or vertex) that is currently used for the load or constraint.
To remove any of the current features, CTRL-click them. If you remove all of the current features, your new selections can be of any type.
- 3 Click the Direction Selection arrow to specify the change of direction using model geometry.



- 4 Click Flip Direction  to reverse the direction, if needed.
- 5 Change any values associated with the load or constraint.
- 6 Click OK to apply the load or constraint changes.

Hide a load symbol



- On the ribbon, click Stress Analysis tab ► Display panel ► Boundary Conditions.
The load symbols are hidden.

Redisplay a load symbol

- On the Stress Analysis tab, click the Boundary Conditions display button again.
The load symbols redisplay.

Temporarily display load location

- In the browser, pause the cursor over the Load or Constraint node. The associated face where the load or constraint is applied highlights.

Change the analysis type

- 1 In the browser, right-click the simulation and click Edit Simulation Properties.
- 2 On the Simulation Properties dialog box, Simulation Type tab, select the new analysis type.

Update Results of Stress Analysis

After you change any of the simulation conditions, or if you edit the part geometry, the current results are invalid. A lightning bolt symbol next to the results node indicates the invalid status. The Update command is located in the node context menu and is enabled.

Update stress analysis results

- Right-click the node that needs an update, and click Update.
New results generate based on your revised solution conditions.

Generate Reports

5

After you run an analysis on a part or assembly, you can generate a report that provides a record of the analysis environment and results.

This chapter tells you how to generate and interpret a report for an analysis, and how to save and distribute the report.

Run Reports

After you run a simulation on a part or assembly, you can save the details of that analysis for future reference. Use the Report command to save all the analysis conditions and results in HTML format for easy viewing and storage.

Generate a report

- 1 Set up and run an analysis for your part.
- 2 Orient the view in the graphics region the way you want to see it in the report.
- 3 On the ribbon, click Stress Analysis tab ► Report panel ► Report to create a report for the current analysis.
- 4 Specify the report parameters in the dialog box. You control the report name, filename, file location, image size, properties reported, and so on. The report generates a variety of image orientations based on the view orientation you established.
- 5 The report is then generated, presented in an internet browser, and saved for your viewing and distribution.

Interpret Reports

The report contains model information, project information, and simulation results.

Model Information

The Model information contains the model name, version of Inventor, and the creation date.

Project Info

The Project Info includes the following:

- Summary, which includes the Author property.
- Project properties, which includes part number, designer, cost, and date created.
- Status property
- Physical properties

Simulation

The simulation section gives details about the simulation conditions.

General objective and settings

This section contains:

- The design objective
- Simulation Type
- Last Modification date
- Setting for Detect and Eliminate Rigid body modes

Advances settings

This section contains:

- Average Element size
- Minimum element size
- Grading Factor
- Maximum Turn Angle
- Create Curved Mesh Elements setting value
- Ignore Small Geometry value

Material(s)

- Material name
- General properties
- Stress properties
- Thermal properties
- Part names, if an assembly report

Operating conditions

- Each force by type and magnitude, with images
- Each constraint by type with images.

Results

- Reaction force and moment on constraints
- Images for each result type as seen in the reports section of the browser

The document path is listed last of all.

Save and Distribute Reports

The report is generated as a set of files to view in a Web browser. It includes the main HTML page, style sheets, generated figures, and other files listed at the end of the report.

Saved Reports

By default reports are saved in the same location as the model being analyzed. The report images are saved in a directory name Images in the same location as the model being analyzed.

Be careful when you name a report. If the file name and location are the same as the previous report, it is possible to overwrite the file without warning. To avoid confusion, it is best to use a different name for each version of a report, or to delete the previous report.

Print Reports

Use your Web browser Print command to print the report as you would any Web page.

Distribute Reports

To make the report available from a Web site, move all the files associated with the report to your Web site. Distribute a URL that points to the report.

Manage Stress Analysis Files

6

Running a stress analysis in Autodesk Inventor® Simulation creates separate files that contain the stress analysis information. In addition, the model file is modified to indicate the presence of the stress files and the name of the files.

This chapter explains how the files are interdependent, and what to do if the files become separated.

Create and Use Analysis Files

After you set up any stress analysis information in Autodesk Inventor Simulation, saving the part or assembly also saves the stress analysis information in the model file. Stress Analysis input and results information, including loads, constraints, and all results are also saved in separate files.

Simulation files are stored in a dedicated folder of the same name as the model file. By default, OLE links are created to each of these files. You can turn off the links by changing the option.

Understand File Relationships

The simulation files are unique to a given model and simulation. Inventor maintains file relationships as needed. There is no reason to work with or modify the simulation files outside of Inventor.

The Save Copy As command copies all Simulation files with the exception of the Mesh and Result data. These must be recomputed for the copied model.

Resolve Missing Files

Under certain circumstances, simulation files can be relocated or missing when working with a model. When you first open a model file, the Resolve Link dialog box displays. You can browse to the location of the simulation files, or you can choose to skip them.

If you skip the files, the Simulation environment can re-compute the files if necessary.

Dynamic Simulation

Part 2 of this manual presents the getting started information for Dynamic Simulation in the Autodesk Inventor® Simulation software. This application environment provides tools to predict dynamic performance and peak stresses before building prototypes.

Get Started with Simulation

7

Autodesk Inventor® Simulation provides tools to simulate and analyze the dynamic characteristics of an assembly in motion under various load conditions. You can also export load conditions at any motion state to Stress Analysis in Autodesk Inventor Simulation to see how parts respond from a structural point of view to dynamic loads at any point in the range of motion of the assembly

About Autodesk Inventor Simulation

The dynamic simulation environment works only with Autodesk Inventor® assembly (.iam) files.

With the dynamic simulation, you can:

- Have the software automatically convert all mate and insert constraints into standard joints.
- Access a large library of motion joints.
- Define external forces and moments.
- Create motion simulations based on position, velocity, acceleration, and torque as functions of time in joints, in addition to external loads.
- Visualize 3D motion using traces.
- Export full output graphing and charts to Microsoft® Excel®.
- Transfer dynamic and static joints and inertial forces to Autodesk Inventor Simulation Stress Analysis or ANSYS Workbench.

- Calculate the force required to keep a dynamic simulation in static equilibrium.
- Convert assembly constraints to motion joints.
- Use friction, damping, stiffness, and elasticity as functions of time when defining joints.
- Use dynamic part motion interactively to apply dynamic force to the jointed simulation.
- Use Inventor Studio to output realistic or illustrative video of your simulation.

Learning Autodesk Inventor Simulation

We assume that you have a working knowledge of the Autodesk Inventor Simulation interface and tools. If you do not, use the integrated Help for access to online documentation and tutorials, and complete the exercises in this manual.

At a minimum, we recommend that you understand how to:

- Use the assembly, part modeling, and sketch environments and browsers.
- Edit a component in place.

We also recommend that you have a working knowledge of Microsoft® Windows® XP or Windows Vista®, and a working knowledge of concepts for stressing and analyzing mechanical assembly designs.

Use Help

As you work, you may need additional information about the task you are performing. The Help system provides detailed concepts, procedures, and reference information about every feature in the Autodesk Inventor Simulation Simulation modules as well as the standard Autodesk Inventor Simulation features.

To access Help, use one of the following methods:

- Click Help ► Help Topics. On the Contents tab, click Dynamic Simulation.
- In any dialog box, click the ? icon.

Understand Simulation Tools

Large and complex moving assemblies coupled with hundreds of articulated moving parts can be simulated. Autodesk Inventor Simulation Simulation provides:

- Interactive, simultaneous, and associative visualization of 3D animations with trajectories; velocity, acceleration, and force vectors; and deformable springs.
- Graphic generation tool for representing and post-processing the simulation output data.

Simulation Assumptions

The dynamic simulation tools provided in Autodesk Inventor Simulation are invaluable in the conception and development steps and in reducing the number of prototypes. However, due to the hypothesis used in the simulation, it provides only an approximation of the behavior seen in real-life mechanisms.

Interpret Simulation Results

Some computations can lead to a misinterpretation of the results or incomplete models that cause unusual behavior. In some cases, a simulation can be impossible to compute. To avoid these situations, be aware of the rules that apply to:

- Relative parameters
- Continuity of laws
- Coherent masses and inertia

Relative Parameters

Autodesk Inventor Simulation Simulation uses relative parameters. For example, the position variables, velocity, and acceleration give a direct description of the motion of a child part according to a parent part through the degree of freedom (DOF) of the joint that links them. As a result, select the initial velocity of a degree of freedom carefully.

Coherent Masses and Inertia

Ensure that the mechanism is well-conditioned. For example, the mass and inertia of the mechanism should be in the same order of magnitude. The most common error is a bad definition of density or volume of the CAD parts.

Continuity of Laws

Numerical computing is sensitive toward discontinuities in imposed laws. Thus, while a velocity law defines a series of linear ramps, the acceleration is necessarily discontinuous. Similarly, when using contact joints, it is better to avoid profiles or outlines with straight edges.

NOTE Using little fillets eases the computation by breaking the edge.

Simulate Motion

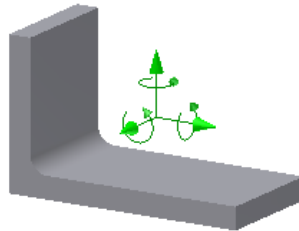
8

With the dynamic simulation or the assembly environment, the intent is to build a functional mechanism. Dynamic simulation adds to that functional mechanism the dynamic, real-world influences of various kinds of loads to create a true kinematic chain.

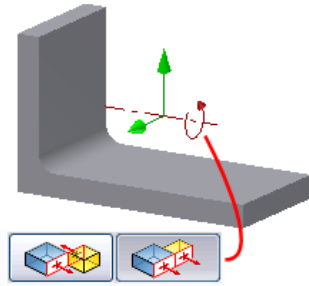
Understand Degrees of Freedom

Though both have to do with creating mechanisms, there are some critical differences between the dynamic simulation and the assembly environment. The most basic and important difference has to do with degrees of freedom.

By default, components in Autodesk Inventor® Simulation have zero degrees of freedom. Unconstrained and ungrounded components in the assembly environment have six degrees of freedom.



In the assembly environment, you add constraints to restrict degrees of freedom.



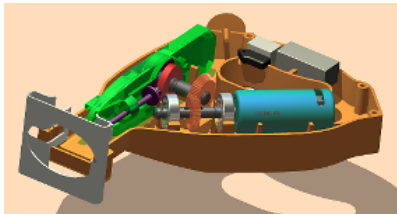
And, in the dynamic simulation environment, you build joints to create degrees of freedom.

Understand Constraints

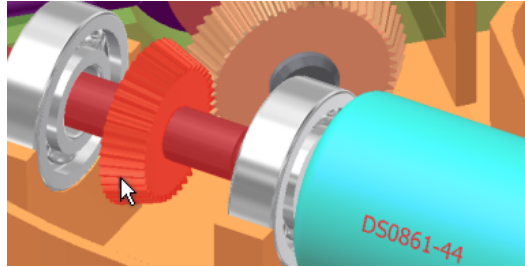
By default, certain constraints that exist in the assembly are automatically converted to joints when dynamic simulation. This eliminates extensive work on your part in creating joints.

NOTE Autodesk Inventor Simulation Simulation converts constraints that have to do with degrees of freedom, such as Mate or Insert, but does not convert constraints that have to do with position, such as Angle.

Open tutorial file



- 1 Set your active project to tutorial_files and then open *Reciprocating Saw.iam*.
- 2 Use the Save As command and give the new file a name such as *RecipSaw-saved.iam*.
- 3 To see how the assembly moves, drag the bevel gear at the end of the motor. The gear revolves but not other components in the kinematic chain.



As you work through the following exercises, save the assembly periodically.

Convert Assembly Constraints

Notice that the assembly moves just as it did in the assembly environment. It seems to contradict preceding explanations, however, the motion you see is borrowed from the assembly environment. Even though you are in Autodesk Inventor Simulation Simulation, you are not yet running a simulation. Since a simulation is not active, the assembly is free to move.

Enter the dynamic simulation environment


- 1 On the ribbon, click Environments tab ► Begin panel ► Dynamic Simulation.

The dynamic simulation environment is activated. If the tutorial dialog box displays, click No.

- 2 In the Simulation Player, click Run .


The Dynamic Simulation browser turns gray and the status slider on the Simulation Player moves, indicating that a simulation is running.

Since we have not created any joints linking the motor to the assembly (and have not specified any driving forces) the assembly does not move.

- 3 If the status slider is still moving, click Stop .

Even though the simulation is not running, the simulation mode is still active. The browser is still grayed.

- 4 Attempt to drag the Bevel Gear component. It does not move.

- 5 On the Simulation Player, click Activate Construction Mode  .
It exits the simulation mode and returns to the Dynamic Simulation construction mode. In construction mode, you perform such tasks as creating joints and applying loads.

Automatically convert assembly constraints

- 1 On the ribbon, click Dynamic Simulation tab ► Manage panel ► Simulation Settings.
This dialog box has the Automatically Convert Constraints to Standard Joints option, which automatically translates certain assembly constraints to standard joints.

When you open an assembly created in Autodesk Inventor Simulation 2010, constraints are automatically converted to joints by default.

NOTE In assemblies created prior to Autodesk Inventor Simulation 2008, Automatically Convert Constraint to Standard Joints is turned off by default. You must go into Dynamic Simulation settings and turn on the option.

- 2 On the Dynamic Simulation Settings dialog box, click Automatically Convert Constraints to Standard Joints to clear the check box. Notice the warning, *all* joints will be deleted as a result of turning this option off.
- 3 Click OK and Apply. All joints are deleted.
- 4 Repeat the process, turning the Automatic Constraint Conversion option back on. The joints are once again created.
- 5 In the Standard Joints folder, notice the standard joints that the software automatically created for you.

There are two bevel gears in this assembly and they work together to transfer motion from the motor to the mechanism that drives the saw blade. Add that motion transfer by adding a rolling joint.

Add a rolling joint

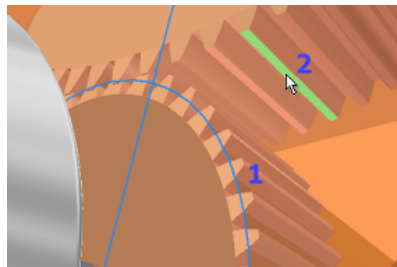
- 1 In the browser, in the Mobile Groups folder, expand the Motor node to expose the Bevel Gear component.
- 2 Right-click the Bevel Gear node and click Edit. You are placed in the Part modeling mode.

- 3 Right-click the Srf1 node and click Visibility. The Bevel Gear construction surface displays. We use this surface to help define the gear relationship.
- 4 At the right end of the ribbon panel, click Return. You are placed back in the simulation environment.

- 5 On the ribbon, click Dynamic Simulation tab ➤ Joint panel ➤ Insert



- 6 In the pull down list, select Rolling: Cone on Cone.
- 7 The Component selector command is active and waiting for input. Select the Pitch Diameter circle at the base of the bevel gear conical surface (1).

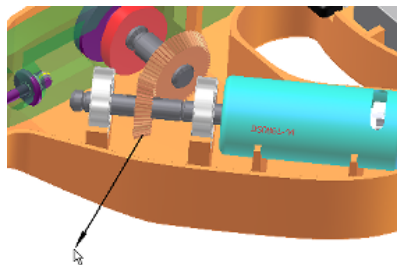


- 8 Automatically, Component selector 2 is active and ready for input. Select a conical face of a tooth on Bevel Gear 2. Do not select an involute face.

NOTE If necessary, expand the Mobile Groups and Cam crank browser nodes to see the gear component.

The new joint is added to the browser below the Standard Joints node.

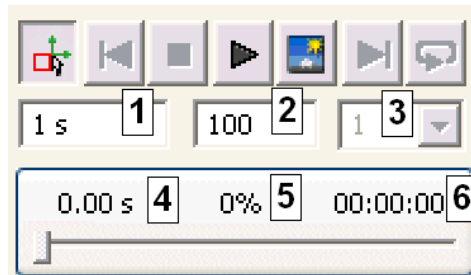
- 9 Click and drag Bevel Gear 1. You can see it move not only Bevel Gear 2, but the entire Cam crank assembly.



Run Simulations

The Simulation Player contains several fields including:

- 1 Final Time
- 2 Images
- 3 Filter
- 4 Simulation Time
- 5 Percent of Realized Simulation
- 6 Real Time of Computation



Simulation Panel

Final Time field	Controls the total time available for simulation.
Images field	Controls the number of image frames available for a simulation.
Filter field	Controls the frame display step. If the value is set to 1, all frames play. If the value is set to 5, every fifth frame displays, and so on. This field is editable when simulation mode is active, but a simulation is not running.
Simulation Time Value	Shows the duration of the motion of the mechanism as would be witnessed with the physical model.
Percent value	Shows the percentage complete of a simulation.
Real Time of Computation value	Shows the actual time it takes to run the simulation. It is affected by the complexity of the model and the resources of your computer.

TIP Click the Screen Refresh command to turn off screen refresh during the simulation. The simulation runs, but there is no graphic representation.

Before you run the simulation, make the following adjustments.

Set up a simulation

- 1 On the Simulation Player, in the Final Time field, enter *0.5 s*.

TIP Use the tooltips to see the names of the fields in the Simulation Player.

- 2 In the Images field, enter *200*. Increasing the image count improves the results when viewed using the Output Grapher.

- 3 On the Simulation Panel, click Run.

As the Motor component moves, the other components making up the kinematic chain respond.

NOTE Because we did not yet specify any frictional or damping forces, the mechanism is lossless. There is no friction automatically created between components.

- 4 If the simulation is still running, on the simulation panel, click the Stop button.

- 5 Click the Activate construction mode button.

As you can see, running the simulation did not result in motion. This is because the kinematic chain is incomplete. In the following chapter you complete the construction and enable motion.

Construct Moving Assemblies

9

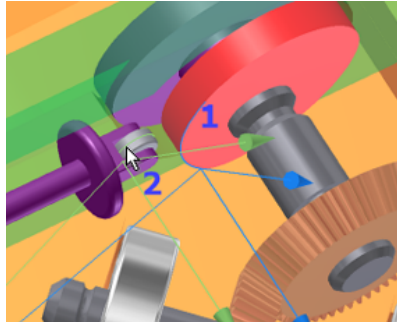
To simulate the dynamic motion in an assembly, define mechanical joints between the parts. This chapter provides basic workflows for constructing joints.

Retain Degrees of Freedom

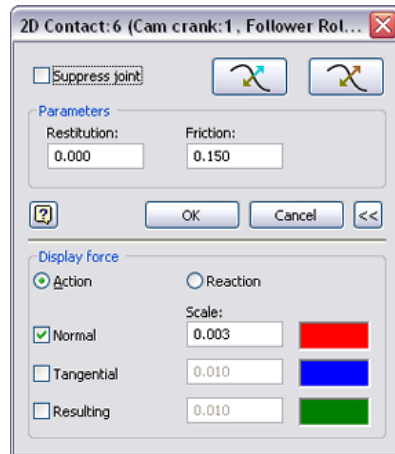
In some cases, it may be appropriate that certain parts move as a rigid body and a joint is not required. As far as the movement of these parts is concerned, the welded body functions like a subassembly moving in a constraint chain within a parent assembly. Similarly, at other times, components making up a welded group will need degrees of freedom for movement within the simulation. Such is the case with the welded group in the Saw model.

Create a 2D contact

- 1 In the browser, expand Mobile Groups.
- 2 Right-click the Follower Roller and click Retain DOF. This allows the roller to retain its motion characteristics.
- 3 In the graphics region, click and drag the Follower away from the Cam crank assembly.
- 4 On the ribbon, click Dynamic Simulation tab ► Joint panel ► Insert Joint and from the list, select 2D Contact.
- 5 Select the Cam profile edge (1).



- 6 Select circular sketch (2) on the roller component.
- 7 Click Apply. As you can see, sketch geometry can be used to help define the simulation.
- 8 Drag the Follower until the roller contacts the cam. Notice it does not penetrate. The 2D contact established a mechanical relationship between the two components.
- 9 Set the properties for the 2D contact and display the force vector. In the browser, right-click the 2D Contact joint and click Properties.

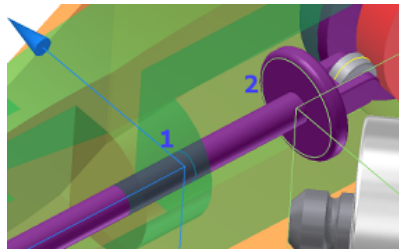


- 10 Set the Restitution values to 0.0.
- 11 Expand the dialog box to access the lower section. Check the Normal box and set the Scale to 0.003.

Add Joints

The Follower is designed to slide through a portion of the Guide component. However, to hold the Follower Roller against the Cam, specify a spring between the Follower and Guide components. Dynamic Simulation has a joint for doing that and more, the Spring/Damper/Jack joint.

- 1 On the ribbon, click Dynamic Simulation tab ► Joint panel ► Insert Joint and in the list, select Spring / Damper / Jack joint.
- 2 On the Guide component, select the hole profile where the follower passes through the Guide (1).
- 3 Select the edge profile where the spring will contact the follower.



The result is a spring joint in the browser and a graphic representation of a spring. The representation is deformable and has action-reaction forces, but does not have mass.



- 4 In the browser, in the Force Joints folder, right-click the Spring joint and click Properties.
- 5 In the main section of the dialog box, set:
 - Stiffness = 2.500 N/mm
 - Free Length = 42 mm

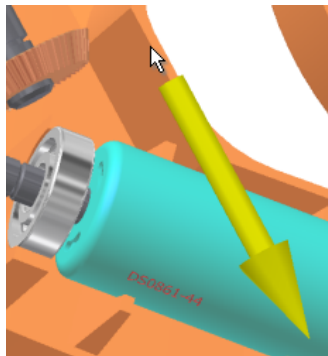
Expand the dialog box and set:

- Radius = 5.2 mm
- Turns = 10
- Wire Radius = .800 mm

6 Click OK. The spring properties and graphical display update.

Define gravity

- 7 In the browser, in the External Loads folder, right-click Gravity and click Define Gravity. Alternatively, you can double-click the Gravity node. If necessary, clear the Suppress check box.
- 8 Select the Case edge, as shown in the following image, to specify a vector for gravity.




Click OK

Impose Motion on Joints

To simulate a saw operation, it is necessary to impose motion. In this case, we apply motion to the motor, just as in the real world case. To impose motion you must edit the joint properties.

- 1 In the browser Standard Joints folder, right-click the Revolution:2 (Saw layout:1. Motor:1) joint and click Properties.
- 2 Click the dof 1 (R) tab.



- 3 Click Edit Joint Motion , and check Enable imposed motion.
- 4 Verify that Velocity is the selected Driving option.
- 5 In the input field, click the arrow to expand the input choices and click Constant Value. Specify 10,000 deg/s
- 6 Click OK.

Run Simulations

Because the simulation is of a high speed device, modify the simulation properties.

TIP Use the tooltips to see the names of the fields on the Simulation Player

Setting simulation options

- 1 On the Simulation Player, Final Time field, enter .5 s, which is sufficient to demonstrate the mechanism.

NOTE The software automatically increases the value in the Images field proportionally to the change in the Final Time field. Press the Tab key to move the cursor out of the Final Time field to update the Images field.

- 2 In the images field, enter 200. Increasing the image count improves the results you view in the Output Grapher.
- 3 Click Run on the Simulation Player.

As the Motor component drives the bevel gear, the remaining parts in the kinematic chain respond.

The direction of gravity has nothing to do with any external notion of "up" or "down," but is set according to the vector you specified.

Also, because we have not yet specified any frictional or damping forces, the mechanism is lossless. There is no friction between components, regardless of how long the simulation runs.

- 4 If the simulation is still running, click the Stop button on the Simulation Player.

Construct Operating Conditions

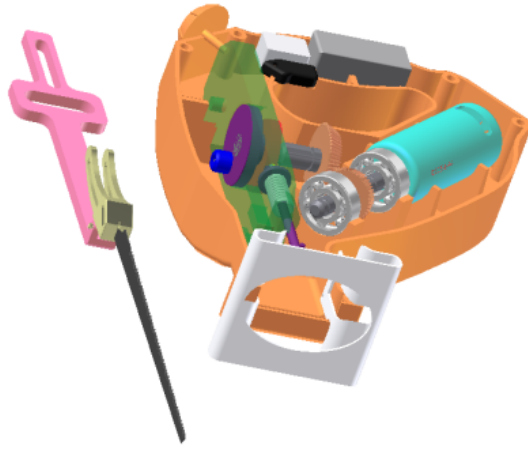
10

This chapter demonstrates how to complete the motion definitions so that the simulation reflects operating conditions.

Complete the Assembly

If the *RecipSaw-saved.iam* assembly is not open, you need to open the file to continue. As you can see, though we have the saw body, we do not have the blade components. To add the blade components it is *not necessary* to leave the simulation environment.:

- 1 Click the Assemble tab to display the assembly ribbon.
- 2 In the Component panel, click Place Component. Select *Blade set.iam* and click Open.
- 3 Position the Blade set assembly near where it will be assembled.



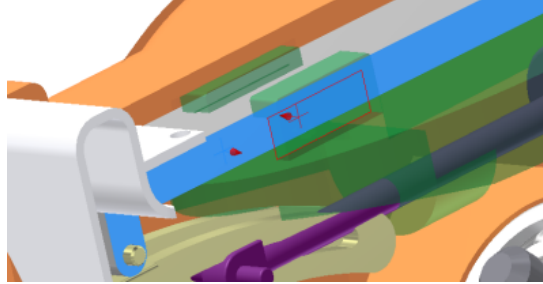
- 4 In the browser, expand the Blade set assembly node to expose the components.
- 5 Select the Scottish Yoke component. On the Quick Access Toolbar, change the color to Chrome.

NOTE If you receive a Design View Representation message about color associativity, select Remove associativity and click OK.

- 6 Add a mate constraint between the Scottish Yoke and Guide to position the yoke on top of the guide.



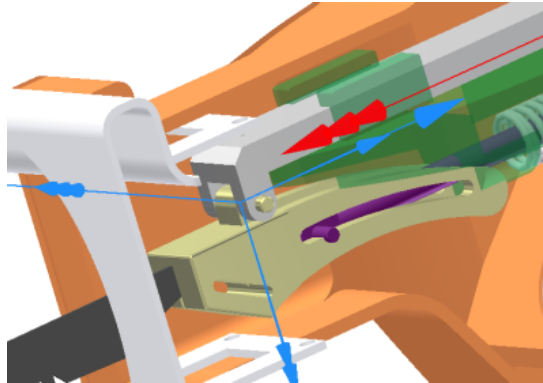
- 7 Add a second mate constraint between the two components to position the yoke within the guide rails. In the browser under Standard Joints, a prismatic joint was created based on adding those constraints.




Add Friction

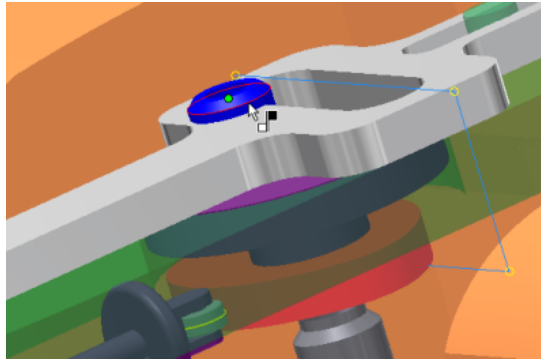
Add friction and complete the yoke-guide relationship

- 1 In the browser, right-click *Blade set.iam* and click Flexible. By setting the assembly to Flexible, the assembly is placed into the welded group folder. Within that assembly the constraints are evaluated and the constraint between the yoke and blade causes the addition of a Revolution joint.



- 2 As was mentioned earlier, the assembly has no friction so far. This step imposes friction on the prismatic joint. Right-click the Prismatic joint for the Guide and Scottish Yoke, and click Properties.

- 3 Click the dof 1 tab. Click the joint forces command . Click Enable joint force. Enter a Dry Friction coefficient of 0.1 and click OK.
- 4 We need to add a constraint to position the Scottish Yoke with respect to the crank assembly. Set the browser view to Model and expand the *Blade set.iam* node.
- 5 Expand the Scottish Yoke node and click the Constraint command.
- 6 In the browser, select Work Plane3 under the Scottish Yoke component.
- 7 In the graphics region, select a circular edge of the Roller component that is part of the Crank cam assembly. A Point-Plane joint is added to reflect the constraint.

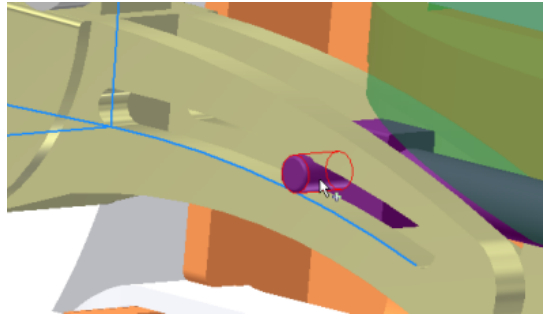


The resulting Point-Plane joint has five degrees of freedom and one constraint. This is enough definition to transfer motion without over constraining the model. Dynamic Simulation detects over-constrained conditions and helps you to resolve them.

Add a Sliding Joint

- 1 The next joint to add is the one between the blade set and the follower, so that the follower travels in the blade clamp.
Before you create that joint, lock the Prismatic Joint between the Guide and Follower components. This prevents the related components from moving and lets the solver work more efficiently.
Right-click the Prismatic:3 (Guide:1, Follower:1) joint and click Lock dofs.

- 2 In the ribbon bar, click the Dynamic Simulation tab to display the simulation commands. Now we'll add the sliding joint.
- 3 In the Joint panel, click Insert Joint. In the pull down list, select Sliding: Cylinder Curve. For input 1 select the blade clamp slot profile on which the follower rides.
- 4 For input 2, select the Follower cylinder face that rides in the slot. Click OK.



- 5 Unlock the prismatic joint.

That completes this chapter on adding components and joints to the assembly. In this chapter you learned to:

- Add assembly components while in the simulation environment.
- Add assembly constraints and see them automatically create standard joints.
- Add joints to simulate mechanical conditions within the assembly.

Index

A

analyses
 meshing 8
 modal 21
 post processing 9
 reports 35
 rerunning on edited designs 31
 results, reading 23, 26
 solving 7
 types, setting 33
 updating 34
 vibration 11
analysis (.ipa) files
 repairing disassociated 40
ANSYS WorkBench 43

B

Boundary Condition command 28
browser, Stress Analysis 13

C

coherent masses and inertia 46
color bar 24
constraints
 assembly 49
 browser display 17
 converting assembly 50
 deleting, adding, and editing 32
continuity of laws 46
contour colors 26

D

deformation results 10, 27
 displaying 29
degree of freedom 47
dialog boxes
 Frequency Options 21

Simulation Panel 52
dynamic simulation 43
 assumptions 45
 coherent masses and inertia 46
 continuity of laws 46
 relative parameters 45
 results 45

E

Element Visibility command 29
equivalent stresses 10, 26
exercises, prerequisites 4

F

factor of safety results 10
files, analysis
 reassociating 40
frequency modes 11
Frequency Options dialog box 21
frequency results options 21

G

geometry, editing 31

H

Help system 4, 44

J

joints 57

K

kinematic chains 47

L

- load symbols 32
 - displaying 28, 33
- loads
 - browser display 17
 - deleting, adding, and editing 32

M

- meshes
 - creating 8
 - displaying 29
- Minimum command 28
- modal analyses 11, 21
- model geometry, editing 31
- modes
 - frequency 11
 - result options 21

N

- natural resonant frequencies 11

O

- Output Grapher 61

P

- panel bar, Stress Analysis 13
- post processing analyses 9
- preprocessing 8
- prerequisites for exercises 4

R

- relative parameters 45
- Report command 35
- reports
 - printing and distributing 38
 - saving 35
- resonant frequency analyses 21
- resonant frequency results 27

results

- animate 27
- deformation 10
- display options 28
- equivalent stresses 10
- frequency options 21
- resonant frequency 27
- reviewing 9
- safety factor 10
- stress analysis, reading 26
- updating 34
- viewing analyses 23
- rigid bodies, creating 55

S

- safety factor results 10, 27
- Simulation Panel 52
 - dialog box 52
 - filter 52
 - final time 52
 - images 52
 - percent value 52
 - real time of computation values 52
 - simulation time value 52
- solutions
 - methods 7
 - rerunning 32
- stress analysis
 - assumptions 8
 - environment 13
 - functionality 6
 - results 26
 - tools 5
- Stress Analysis Update command 34
- stresses, equivalent 10

T

- tools, stress analysis 13

U

- update analyses 34

V

vibration frequency analyses 21
von Mises stress 10

W

welded bodies 55
workflows
 running modal analyses 21

