

Multibody Dynamics Module

User's Guide

Multibody Dynamics Module User's Guide

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023801

C o n t e n t s

Chapter 1: Introduction

About the Multibody Dynamics Module	10
What Can the Multibody Dynamics Module Do?	10
Where Do I Access the Documentation and Application Libraries?	11
Overview of the User's Guide	15

Chapter 2: Multibody Dynamics Modeling

The Multibody Dynamics Model	18
Rigid and Flexible Parts	18
Meshing	18
Connecting Parts	19
Gears	19
Connecting to Other Physics Interfaces	20
Joints	21
About Joints	21
Prismatic Joint	30
Hinge Joint	31
Cylindrical Joint	31
Screw Joint	33
Planar Joint	34
Ball Joint	35
Slot Joint	35
Reduced Slot Joint	36
Fixed Joint	37
Distance Joint	37
Universal Joint	37

Attachments	39
Cam-Follower	42
Springs and Dampers	43
Gears	44
About Gears	44
About Gear Pairs.	46
Gear Geometries	50
Gear Tooth Geometry (External: 2D)	51
Gear Tooth Geometry (Internal: 2D)	51
Rack Tooth Geometry (2D)	52
Spur Gear Geometry (External: 2D)	53
Spur Gear Geometry (Internal: 2D)	53
Spur Rack Geometry (2D)	54
Spur Gear Geometry (External: 3D)	55
Spur Gear Geometry (Internal: 3D)	55
Helical Gear Geometry (External: 3D)	56
Helical Gear Geometry (Internal: 3D)	57
Bevel Gear Geometry (3D)	58
Worm Gear Geometry (3D)	59
Spur Rack Geometry (3D)	60
Helical Rack Geometry (3D)	61
Bearings	63
Bearing Modeling	63
Initialization in Time-Dependent Studies	65
Geometric Nonlinearity	66
Multiphysics Couplings	67
The Fluid-Multibody Interaction Interface	68
The Fluid-Multibody Interaction, Assembly Interface	71

Chapter 3: Multibody Dynamics Theory

Joint Theory	76
Representation of Rotations	76
Prismatic Joint Theory	78
Hinge Joint Theory	82
Cylindrical Joint Theory	84
Screw Joint Theory	86
Planar Joint Theory	87
Ball Joint Theory	88
Slot Joint Theory	89
Reduced Slot Joint Theory	91
Fixed Joint Theory	93
Distance Joint Theory	93
Universal Joint Theory	94
Theory for Joint Constraints	97
Theory for Joint Locking	103
Theory for Springs and Dampers on Joints	106
Theory for Prescribed Motion on Joints	108
Theory for Applied Forces and Moments on Joints	110
Theory for Friction in Joints	111
Theory for Joint Forces and Moments Computation	118
Cam-Follower Theory	121
Cam	121
Follower	121
Closest Point Determination	121
Cam with Boundary Selection	122
Cam with Edge Selection	123
Cam-Follower Connection Force Computation	124
Spring-Damper Theory	126
Connection Points and Spring Length	126
Deactivation	127
Spring and Damping Forces	127
Spring and Damping Energies	128

Gear Theory	130
Spur Gear Theory	130
Helical Gear Theory	132
Bevel Gear Theory	134
Worm Gear Theory	135
Spur Rack Theory	136
Helical Rack Theory.	138
Gear Pair Theory	139
Theory for Gear Pairs	139
Theory for Worm and Wheel	147
Theory for Rack and Pinion	149
Theory for Gear Elasticity	152
Theory for Transmission Error	159
Theory for Backlash.	160
Theory for Gear Friction.	162

Chapter 4: Multibody Dynamics

The Multibody Dynamics Interface	166
Domain, Boundary, Edge, Point, and Pair Nodes for the Multibody	
Dynamics Interface.	171
Initial Values	175
Initially Rigid	176
Attachment	177
Rigid Domain	179
Center of Rotation Nodes	181
Initial Values (Rigid Domain)	182
Prismatic Joint	183
Hinge Joint	185
Cylindrical Joint	186
Screw Joint	187
Planar Joint	188
Ball Joint	188
Slot Joint	189
Reduced Slot Joint	190

Fixed Joint	191
Distance Joint	192
Universal Joint	193
Joint Elasticity	195
Constraints	197
Locking	199
Spring and Damper	202
Prescribed Motion	204
Applied Force and Moment	206
Friction	207
Contact Area	210
Joint Axis	210
Input Shaft Orientation	210
Output Shaft Orientation	211
Center of Joint Nodes	211
Source Point Nodes	212
Destination Point Nodes	212
Source Filter	213
Destination Filter	213
Spring-Damper	214
Source Point (for Spring-Damper)	215
Destination Point (for Spring-Damper)	216
Base Motion	216
Cam-Follower	217
Spur Gear	220
Helical Gear	223
Bevel Gear	224
Worm Gear	225
Spur Rack	226
Helical Rack	227
Gear Axis	228
Rack Axis	228
Gear Pair	229
Worm and Wheel	231
Rack and Pinion	232
Gear Elasticity	233
Transmission Error	235
Backlash	237

Friction (Gear)	238
Radial Roller Bearing	239
Flexible Foundation	241
Moving Foundation	242

Chapter 5: Lumped Mechanical System

The Lumped Mechanical System Interface	244
Nodes for the Lumped Mechanical System Interface	245
Mass	246
Spring	247
Damper	248
Impedance	249
Displacement Source	250
Force Source	251
External Source	252
Subsystem Definition	253
Subsystem Instance	254
Fixed Node	254
Free Node	255
Displacement Node.	255
Velocity Node	256
Acceleration Node	257
Force Node.	257
Impedance Node.	258
Theory for the Lumped Mechanical System Interface	259
Dependent Variables	259
Theory for Two-Port Components	259
Theory for Single-Port Components	265
Index	271

Introduction

This guide describes the Multibody Dynamics Module, an optional add-on package for the COMSOL Multiphysics® simulation software designed to perform static or dynamic analysis for a system of flexible and rigid components connected through a set of joints, springs, dampers, gears, and cams.

This chapter introduces you to the capabilities of this module. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to the chapters in this guide.

In this chapter:

- [About the Multibody Dynamics Module](#)
- [Overview of the User's Guide](#)

About the Multibody Dynamics Module

In this section:

- [What Can the Multibody Dynamics Module Do?](#)
- [Where Do I Access the Documentation and Application Libraries?](#)

What Can the Multibody Dynamics Module Do?

The Multibody Dynamics Module, which requires the Structural Mechanics Module, can be used to model the static and dynamic behavior of rigid or flexible components connected using joints with certain degrees of freedom. The components can then undergo complex combinations of translational and rotational motions with respect to one another. Some major application areas are for automotive applications, aerospace engineering, biomechanics, biomedical instruments, robotics, and general dynamic simulations.

The main components of this module are [The Multibody Dynamics Interface](#) and [The Lumped Mechanical System Interface](#).

The Multibody Dynamics interface is used to model assemblies of flexible components, rigid components, or the combination of both. Flexible components can be defined through solid, shell, or beam elements. They can also have geometric and material nonlinearity. With this physics interface, you can simultaneously perform a system analysis and a detailed component analysis of a mechanical system.

The features that are used to model connections are joints, attachments, rigid domains, springs and dampers. The joints that connect the different structural parts can have many different properties, including elastic connections, viscous damping, friction losses, stop conditions, and lock conditions.

There are several types of joints available for modeling different types of connections between components. Attachments are used by joints to establish a connection primarily between flexible components, but can also be used with rigid bodies. A rigid domain models a rigid component, and it is directly used in a joint to establish its connection with another rigid or a flexible component. Elastic springs and viscous dampers can also be used to connect rigid or flexible components.

This interface is also used to model geared systems for power transmission. There are several types of gears available to connect and transfer power from one shaft to another. The gear pairs that connect two gears can have many different properties, including mesh stiffness and damping, static transmission error, backlash, and friction losses. The results obtained from the gear dynamics can be further used for the fatigue analysis of the gear or for the acoustic analysis to find out the noise emitted by the system.

The Multibody Dynamics interface can be coupled to a heat transfer or acoustics interface through multiphysics coupling nodes.

The Lumped Mechanical System interface is intended to model displacements and forces in a mechanical system using a circuit representation with lumped components such as masses, springs, and dampers. In addition to the displacement and force sources in the system, terminal nodes are used to define forces, displacements, velocities, accelerations, or impedances as boundary conditions.



In the *COMSOL Multiphysics Reference Manual*:

- [Studies and Solvers](#)
- [The Physics Interfaces](#)
- For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see [Physics Interface Guide](#).

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.







If you are reading the documentation as a PDF file on your computer, the [blue links](#) do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.




THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.



Opening Topic-Based Help

The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (). This is called *topic-based* (or *context*) *help*.

	<p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder, Application Builder, or Physics Builder click a node or window and then press F1.• On any toolbar (for example, Home, Definitions, or Geometry), hover the mouse over a button (for example, Add Physics or Build All) and then press F1.• From the File menu, click Help ().• In the upper-right corner of the COMSOL Desktop, click the Help () button.
---	--


 	<p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder or Physics Builder click a node or window and then press F1.• On the main toolbar, click the Help () button.• From the main menu, select Help>Help.
--	---

Opening the Documentation Window

	<p>To open the Documentation window:</p> <ul style="list-style-type: none">• Press Ctrl+F1.• From the File menu select Help>Documentation ().
---	---



To open the **Documentation** window:

- Press Ctrl+F1.
- On the main toolbar, click the **Documentation** () button.
- From the main menu, select **Help>Documentation**.



THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or app. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.



[The Application Libraries Window](#) in the *COMSOL Multiphysics Reference Manual*.


Opening the Application Libraries Window

To open the **Application Libraries** window ():



- From the **Home** toolbar, **Windows** menu, click () **Applications Libraries**.


- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select () **Update COMSOL Application Library**.



Select **Application Libraries** from the main **File>** or **Windows>** menus.



To include the latest versions of model examples, from the **Help** menu select () **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *Multibody Dynamics Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions how to use COMSOL Multiphysics in general are included with the *COMSOL Multiphysics Reference Manual*.



As detailed in the section [Where Do I Access the Documentation and Application Libraries?](#) this information can also be searched from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS AND INDEX

To help you navigate through this guide, see the [Contents](#) and [Index](#).

MULTIBODY DYNAMICS MODELING

The [Multibody Dynamics Modeling](#) chapter outlines the modeling guidelines for the Multibody Dynamics interface. The [The Multibody Dynamics Model](#) section describes various elements of a multibody system. The [Joints](#) section describes the types of joints and the additional functionality available. The [Attachments](#) section describes the use and formulation of attachment features. The [Cam-Follower](#) section describes the formulation of cam-follower connection. The [Springs and Dampers](#) section describes the formulation of an elastic spring and a viscous damper. The [Gears](#) section describes the type of gears, gear pairs, and the additional functionality available for modeling of gears. The [Gear Geometries](#) section describes the parametrized gear geometries available. The section on [Initialization in Time-Dependent Studies](#) describes the consistent initialization in a multibody system. The [Geometric Nonlinearity](#) section talks about the cases when a geometric nonlinear analysis is required. In the end, [Multiphysics Couplings](#) section describes all the built-in multiphysics coupling features available for multibody dynamics.

THE THEORY OF THE MULTIBODY DYNAMICS INTERFACE

The [Multibody Dynamics Theory](#) chapter outlines the theory for the Multibody Dynamics interface. The [Joint Theory](#) section describes the types of joints and the additional functionality available. The [Cam-Follower Theory](#) section describes the theory for the cam and follower connection. The [Spring-Damper Theory](#) section describes the formulation of an elastic spring and a viscous damper. The [Gear Theory](#)

section describes the theory of all the gear types. In the end, the [Gear Pair Theory](#) section describes the theory of all the gear pairs.

THE MULTIBODY DYNAMICS INTERFACE

The [Multibody Dynamics](#) chapter describes all the physics nodes available in [The Multibody Dynamics Interface](#). Joints, spring-dampers, cam-followers, attachments, rigid domains, base motions, initial values, gears, and gear pairs are described in detail. The rest of the physics nodes are described in the *Structural Mechanics Module User's Guide*.

LUMPED MECHANICAL SYSTEM

The [Lumped Mechanical System](#) chapter describes all the physics nodes available in [The Lumped Mechanical System Interface](#). The corresponding theory for all the physics nodes is described in the [Theory for the Lumped Mechanical System Interface](#) section.

Multibody Dynamics Modeling

This chapter introduces you to the theory for the Multibody Dynamics interface features.

In this chapter:

- [The Multibody Dynamics Model](#)
- [Joints](#)
- [Attachments](#)
- [Cam-Follower](#)
- [Springs and Dampers](#)
- [Gears](#)
- [Gear Geometries](#)
- [Bearings](#)
- [Initialization in Time-Dependent Studies](#)
- [Geometric Nonlinearity](#)
- [Multiphysics Couplings](#)

The Multibody Dynamics Model

Rigid and Flexible Parts

One of the big advantages of the Multibody Dynamics Module is the ease by which you can mix rigid and flexible parts. Usually, all or the majority of the parts in a multibody simulation are rigid and are thus represented by only the degrees of freedom of a rigid body. The material model for a rigid part is **Rigid Domain**.

Sometimes, however, you may want to include the flexibility of one or several domains. To do that, use a **Linear Elastic Material**. This is the default material model in the Multibody Dynamics interface, so the workflow is to add as many **Rigid Domain** nodes as needed and assign domains to them. The remaining domains will then be linear elastic and thus flexible. By adding a Solid Mechanics interface, you can get access to many other material models for the flexible domains.

Meshing

A rigid domain is only represented by a set of ODEs, representing the equations of motion for a rigid body. The mesh on a rigid domain is only used for integrating properties like mass, center of gravity, and moments of inertia. This means that the mesh can be much coarser than what is usually the case in a finite element model. Since there is still a computational and memory consumption cost involved with each element, you should strive for a very coarse mesh in rigid domains.

When meshing a flexible body, you have to make the same considerations as when meshing any other finite element model.



In the Multibody Dynamics interface, the default shape function order for the flexible parts is linear, as opposed to the Solid Mechanics interface where it is quadratic. The purpose is to keep down the number of degrees of freedom in those cases where the stiffness is more important than stress evaluation.

If you need more accurate stress results, you can either use a finer mesh, or change the discretization to **Quadratic serendipity** or **Quadratic** in the **Discretization** section settings of the interface.

Connecting Parts

The most important feature for connecting part is a *joint*. A joint will in general allow relative movement between two parts in one or more degrees of freedom. This is described in detail in the [Joints](#) section.

It is also possible to connect parts by springs and dampers, as described in the [Springs and Dampers](#) section.

A part can be connected to several joints, springs, or dampers. The way to attach them differs between a rigid and a flexible object.

- A flexible body must be connected through an *attachment*, as described in the [Attachments](#) section. Add an **Attachment** node to the model and select the attached boundaries. The attachment is then referenced in, for example, a joint.
- A rigid domain can be connected directly, by just specifying the location of the connection. The connection point does not even have to physically inside the rigid domain since the domain can be considered as an abstract object.
- A rigid domain can also be connected by an attachment. This is convenient if you consider switching between a rigid and a flexible formulation for this domain.



A Rigid Domain can consist of a selection of several geometrical domains. These domains will act as a single rigid object, irrespective of whether they are geometrically connected or not.

Adjacent domains selected in different Rigid Domains nodes are independent objects, and can even penetrate each other.

Gears

A gear is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the gear pair node while connecting to other gears. Similar to the rigid domain, the gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

Connecting to Other Physics Interfaces

When modeling flexible parts, it is possible to add other structural mechanics interfaces to get additional functionality:

- The Solid Mechanics interface has several features which are not available directly from the Multibody Dynamics interface.
- If you have the Nonlinear Structural Mechanics Module or the Geomechanics Module, then you can get access to the whole range of material model by adding a Solid Mechanics interface. In this way, you can for example include a hyperelastic or elastoplastic domain in your multibody dynamics simulation.
- To model thin or slender flexible objects, you can add a Shell or Beam interface.

There are two ways in which you can connect to objects from the other structural mechanics interfaces:

- You can connect through joints or springs. In the Solid Mechanics, Shell, or Beam interfaces, you can add **Attachment** nodes. Such attachments will then appear in the lists of attachments in various features in the Multibody Dynamics interface.
- You can also connect to objects in the Shell and Beam interfaces through the **Solid-Shell Connection** and **Solid-Beam Connection** multiphysics couplings.

Joints

About Joints

A joint is used to allow a certain type of relative motion between the two components it connects. A joint is a global feature that needs two *attachments*, one from each component. One attachment acts as a *source* and the other as a *destination*.



The two attachments belonging to a joint can be [Attachment](#) features, **Rigid Domain** features, or a combination of the two. The source attachment can also be set to be either *fixed* or to have a given **Base motion**. In the case of a fixed attachment, it cannot translate or rotate. This avoids the modeling of extra fixed rigid domains when a structure is “grounded”. The base motion is a generalization of the fixed attachment, where the environment to which it is attached can have a prescribed displacement, velocity, or acceleration.

The source attachment moves independently in space, whereas the destination attachment is constrained to follow the source attachment allowing for some relative degrees of freedom in the form of translation or rotation. The relative degrees of freedom are determined by the type of joint. In each joint, the relative degrees of freedom are represented by ODE variables.

The degrees of freedom that are constrained in a joint can be rigidly constrained or contain some relative elastic displacement.

The rest of this section has these topics:

- Joint Types and Degrees of Freedom
- Selecting Attachments
- Joint Axis and Local Coordinate System
- Positioning the Joint Center
- Constraints
- Locking
- Springs and Dampers on Joints
- Elasticity in Joints
- Prescribed Motion
- Applied Forces and Moments on Joints
- Friction
- Computing Joint Forces and Moments
- Initial Values of Joint Degrees of Freedom

JOINT TYPES AND DEGREES OF FREEDOM

Table 2-1 lists the joint types for 3D models based on the possible relative degrees of freedom.

TABLE 2-1: JOINT TYPES AND DEGREES OF FREEDOM IN 3D

JOINT TYPE	TOTAL DEGREES OF FREEDOM	TRANSLATIONAL DEGREES OF FREEDOM	ROTATIONAL DEGREES OF FREEDOM
Prismatic Joint	1	1	0
Hinge Joint	1	0	1
Cylindrical Joint	2	1	1
Screw Joint*	1*	1	1
Planar Joint	3	2	1
Ball Joint	3	0	3
Slot Joint	4	1	3
Reduced Slot Joint	2	1	1
Fixed Joint	0	0	0
Distance Joint**	5	2**	3
Universal Joint	2	0	2

* For a Screw Joint, the two degrees of freedom are linearly dependent.

** For a Distance joint, all three translational degrees for freedom are not independent.

Table 2-2 lists the available joint features for 2D models:

TABLE 2-2: JOINT TYPES AND DEGREES OF FREEDOM IN 2D

JOINT TYPE	TOTAL DEGREES OF FREEDOM	TRANSLATIONAL DEGREES OF FREEDOM	ROTATIONAL DEGREES OF FREEDOM
Prismatic Joint	1	1	0
Hinge Joint	1	0	1
Reduced Slot Joint	2	1	1
Fixed Joint	0	0	0
Distance Joint**	2	1**	1

** For a Distance joint, both translational degrees for freedom are not independent.

SELECTING ATTACHMENTS

There are no rules for how to select the source or destination attachment for a joint, but one selection can be more convenient than the other. The only difference is that the interpretation of certain quantities changes if the source and destination are swapped. Some of the important quantities and the relationship with the source and destination attachments are:

- The joint coordinate system follows the source attachment.
- All the relative degrees of freedom are created on the destination attachment.
- The joint moments are computed around the center of joint attached to the source attachment.
- In the [Ball Joint](#) and [Slot Joint](#), the inclination angle is the angle between the reference axis and the destination axis. The axial rotation occurs about the destination axis. This reference axis is defined on the source attachment.
- In the [Planar Joint](#), all axes in its subfeatures follow the source attachment.

POSITIONING THE JOINT CENTER

You can position the center of a joint in different ways:

- At the centroid of the source attachment.
- At the centroid of the destination attachment.

- At the centroid of some selected entities (boundaries, edges, or points). As a special case a single point can be used.
- Explicitly given coordinates.



Attachments must be used for flexible components, and are optional for rigid components. The first two options are only available if an attachment is selected as source or destination.

JOINT AXIS AND LOCAL COORDINATE SYSTEM

The local coordinate system of a joint, which can be seen as rigidly connected to the source attachment, consists of three axes: joint axis (\mathbf{e}_1), second axis (\mathbf{e}_2), and third axis (\mathbf{e}_3). You specify the initial direction joint axis (\mathbf{e}_{10}), using one of the following methods:

- **Method 1:** Give the components explicitly in a selected coordinate system.
- **Method 2:** Select one of the axes of a selected coordinate system.
- **Method 3:** Select an edge that is parallel to the joint axis.

For all methods, the axis is normalized to give a unit vector.

CONSTRAINTS

Constraints are used to restrict the relative motion between the two components sharing a joint. You can specify the upper and lower bound of the relative motion. Applying constraints on the relative motion can be interpreted as placing an imaginary stopper that restricts the motion of the destination attachment with respect to the source attachment in a prescribed direction.



[Theory for Joint Constraints](#)

LOCKING

The **Locking** feature is used to lock the relative motion between the two components connected to a joint. You specify limits on the relative motion in the form of an upper and lower bound. This can be interpreted as an imaginary object (like a snap hook) that locks one of the free relative degrees of freedom of the destination attachment when it reaches the limiting value.

The Locking feature differs from [Constraints](#) in that the value of the particular joint degree of freedom becomes constant for the rest of the analysis once it attains its prescribed limiting value.



Theory for Joint Locking

SPRINGS AND DAMPERS ON JOINTS

You can use the [Spring and Damper](#) feature to apply a spring or a dashpot to act on the relative motion in a joint. A dashpot can be used to account for various kinds of losses at the joint. The spring can be given a predeformation so that the spring equilibrium position is not the same as the initial state.



Theory for Springs and Dampers on Joints

ELASTICITY IN JOINTS

In the directions in which relative motion is not allowed in a joint, the default case is that the source and destination attachments are rigidly connected. It is possible to instead insert an elastic connection by setting **Joint Elasticity** to **Elastic joint** in the settings for the joint. There are several occasions when you may want to use this option:

- There is a physical flexibility in the joint, like a bushing.
- The mechanical system is overconstrained by the chosen set of joints. You can then release some degrees of freedom by making them elastic. This should typically be done in directions where no significant forces act.
- You have a mechanism which is not possible to model by the built-in joints. You can then add degrees of freedom to the built-in joint by making them elastic, and not supply any stiffness.
- The joint gets deactivated and the two bodies are no longer connected after certain time in a transient simulation. You then can use the elastic joints for this purpose by making the stiffness and damping properties a function of time.

In addition to the elasticity, you can also add viscous damping to a degree of freedom selected as being elastic.



The theory for joint elasticity is described separately for each joint type:

- [Joint Elasticity in Prismatic Joint](#)
- [Joint Elasticity in Hinge Joint](#)
- [Joint Elasticity in Cylindrical Joint](#)
- [Joint Elasticity in Planar Joint](#)
- [Joint Elasticity in Ball Joint](#)
- [Joint Elasticity in Slot Joint](#)
- [Joint Elasticity in Reduced Slot Joint](#)
- [Joint Elasticity in Fixed Joint](#)
- [Joint Elasticity in Universal Joint](#)

PRESCRIBED MOTION

With [Prescribed Motion](#) you can control the relative motion in a joint, either by prescribing the displacement or the velocity. This is useful when the degrees of freedom in the joint are not free but are a known function of time. This is common in the field of robotics, for example.



Evaluating reaction force for the prescribed motion by selecting **Evaluate reaction forces** in the **Reaction force settings** section of the settings for [Prescribed Motion](#) has several benefits over evaluating it using joint forces and moments computed using weak constraints:

- It includes the contributions of springs, dampers, and friction applied on the joint.
- It works even if there are redundant constraints in the system, especially in a rigid body system, because joint constraints cannot be applied in the weak form in such a system.
- It is computationally more efficient as it adds fewer degrees of freedom.



[Theory for Prescribed Motion on Joints](#)

APPLIED FORCES AND MOMENTS ON JOINTS

The [Applied Force and Moment](#) feature is used to apply forces and moments on all types of joints. It can be applied at the source or destination joint attachment, or directly to the joint degrees of freedom.



Theory for Applied Forces and Moments on Joints

FRICITION

The [Friction](#) feature is used to add frictional loss to a joint. It is available for [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Ball Joint](#).

The friction force is modeled using a continuous friction law, which is capable of modeling sliding-sticking phenomena. A strict application of Coulomb's law involves discrete transition from sticking to sliding and vice versa, as dictated by a vanishing relative velocity. These discrete transitions cause numerical difficulties and to avoid them, the friction force is approximated with a continuous friction law:

$$F_f = -\mu N \frac{v}{|v|} \left(1 - e^{-\frac{|v|}{v_0}} \right)$$

where F_f is the friction force, μ is the frictional coefficient, N is the normal force, v is the slip velocity, and v_0 is the characteristic slip velocity. The term

$$\left(1 - e^{-\frac{|v|}{v_0}} \right)$$

is called the *regularization factor*.

The regularization factor is used to smoothen the friction force discontinuity. The characteristic slip velocity should be chosen to be small in comparison to the characteristic relative velocities encountered during the simulation. The continuous friction law describes both sliding and sticking behavior; that is, it completely replaces Coulomb's law. Sticking is replaced by creeping between the contacting bodies with a small relative velocity.



Theory for Friction in Joints

COMPUTING JOINT FORCES AND MOMENTS

It is often of interest to evaluate the internal forces (the reaction forces) in a joint. There are three ways to compute forces and moments — [Summing Reaction Forces Over the Boundaries](#), or [Using Weak Constraints](#), or [Using Penalty Formulation](#).

The joint forces and moments are evaluated in the global spatial coordinate system as well as in a joint local coordinate system. The joint forces and moments are referred to the center of joint in the current position. For the joints that have translational degrees of freedom, the center of joint should be interpreted as fixed to the source side.

When a joint connects two rigid bodies, the default is not to compute the joint forces. The reason is that it is common that rigid body systems are overconstrained, and then numerical problems can occur.

Summing Reaction Forces Over the Boundaries

This is the default method for computing the joint forces and moments for elastic objects. The computation can be done on either the source or destination attachment. The convention is that the force and moment are the ones acting on the attachment from the joint. The sign changes if you switch between using the source or destination attachment for the evaluation.



To use this method at least one of the components connected to the joint must be flexible.

Using Weak Constraints

When using weak constraints, the joint constraints are applied in a weak form, and the values of the Lagrange multipliers give the joint forces and moments. This works for both flexible and rigid components. The use of Lagrange multipliers can, however, have an effect on the structure of the equation system, which limits the solvers that can be used.

The sign convention in this case is that the forces and moments should be interpreted as acting on the destination attachment.



Using the weak constraint option in an overconstrained rigid body system often leads to numerical difficulties.

Using Penalty Formulation

When using the penalty formulation, the constraints in the joint are implemented as stiff springs. You have to provide a *penalty factor*, which essentially is the spring constant. The penalty method will thus not fulfill the constraints exactly, but it has the advantage of making the model less sensitive to overconstraints.



Theory for Joint Forces and Moments Computation

INITIAL VALUES OF JOINT DEGREES OF FREEDOM

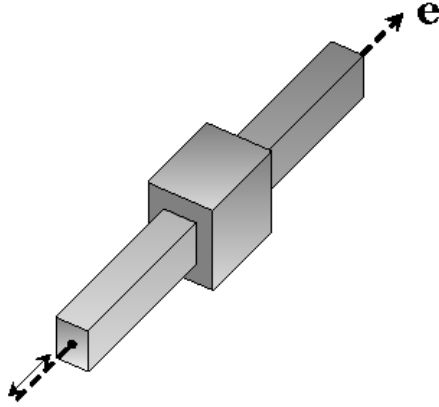
You do not need to supply separate initial values for joint degrees of freedom. They are consistently initialized from the degrees of freedom for a material model, either a displacement field in a flexible component or the ODE degrees of freedom in a rigid component.



In cases where locking and constraint conditions are present on a joint, the initial values of the joint degrees of freedom should be within the specified minimum and maximum limits.

Prismatic Joint

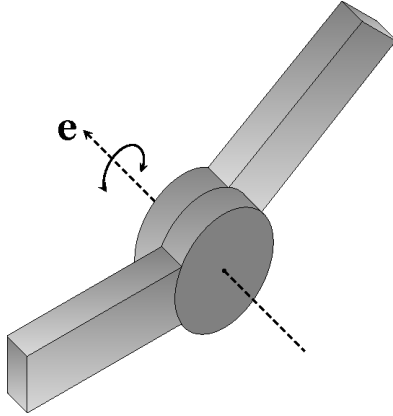
The **Prismatic Joint**, also known as a translational or sliding joint, has one translational degree of freedom between the two connected components. The components are free to translate, or slide, relative to each other along the joint axis.



Prismatic Joint Theory

Hinge Joint

The [Hinge Joint](#), also known as a revolute joint, has one rotational degree of freedom between the two components. The two components are free to rotate relative to each other about the axis of the joint.

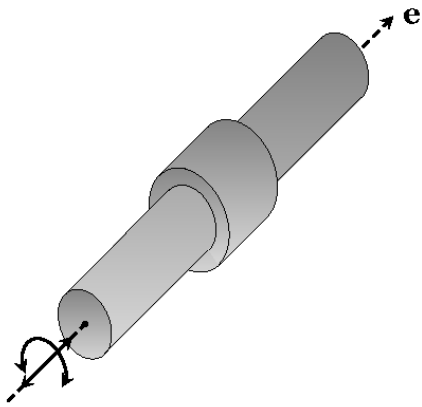


[Hinge Joint Theory](#)

Cylindrical Joint

The [Cylindrical Joint](#) has one translational and one rotational degree of freedom between the two connected components. The components are free to slide and rotate

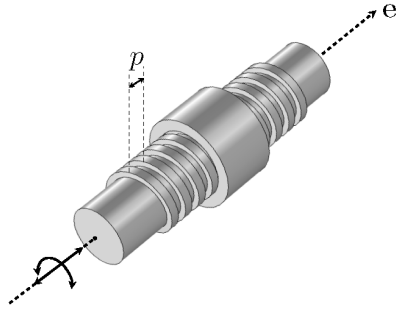
relative to each other along the axis of joint. A cylindrical joint can be thought of as a combination of a prismatic joint and a hinge joint.



Cylindrical Joint Theory

Screw Joint

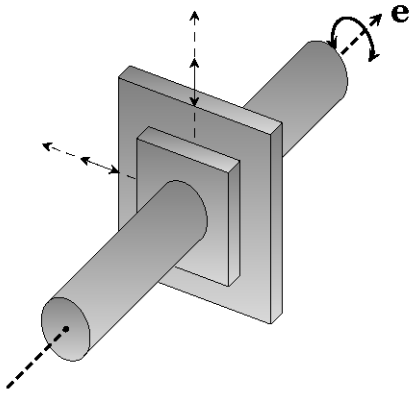
The **Screw Joint** is similar to the cylindrical joint except that the relative displacement and the relative rotation, instead of being independent, are linearly related to each other. Hence, this joint effectively has only one degree of freedom.



Screw Joint Theory

Planar Joint

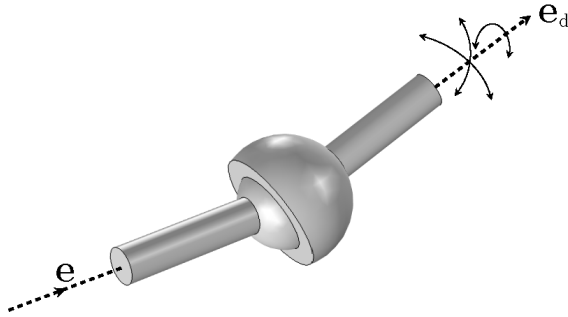
The **Planar Joint** has two translational and one rotational degrees of freedom between the two components. Both components are free to translate relative to each other in a plane perpendicular to the axis of joint, and they are also free to rotate about this axis.



Planar Joint Theory

Ball Joint

The **Ball Joint**, also known as a spherical joint, has three rotational degrees of freedom between the two attached components. The two components are free to rotate relative to each other along all three axes.

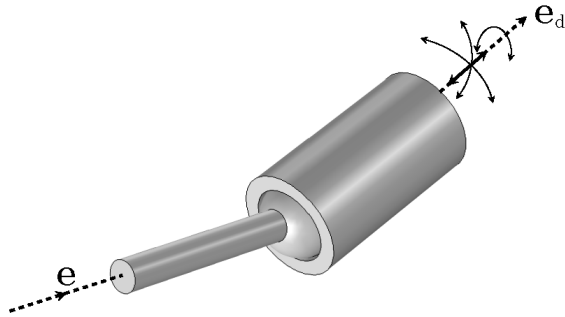


Ball Joint Theory

Slot Joint

The **Slot Joint** has three rotational and one translational degrees of freedom between the two attached components. The components are free to rotate relative to each other

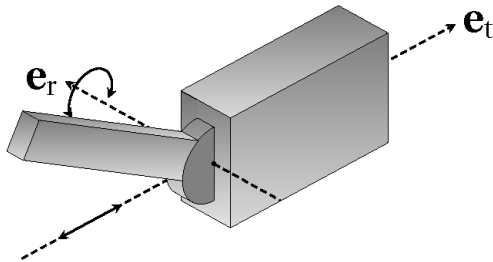
about all three axes and also free to translate along the joint translation axis. The slot joint can be seen as a combination of the prismatic joint and the ball joint.



Slot Joint Theory

Reduced Slot Joint

The **Reduced Slot Joint** has one rotational and one translational degree of freedom between the two attached components. Both components of the system are free to rotate about the joint rotational axis and free to translate along the joint translation axis. The reduced slot joint can be thought of as a combination of the prismatic joint and the hinge joint, with different axes for the allowed degrees of freedom. The translational axes and the rotational axis are orthogonal to each other.



Reduced Slot Joint Theory

The reduced slot joint in 2D is similar to the reduced slot joint in 3D. The only difference is the initial joint rotational axis ($\mathbf{e}_{\text{rot}0}$), which does not change in 2D and is always the out-of-plane axis (\mathbf{e}_z).

Fixed Joint

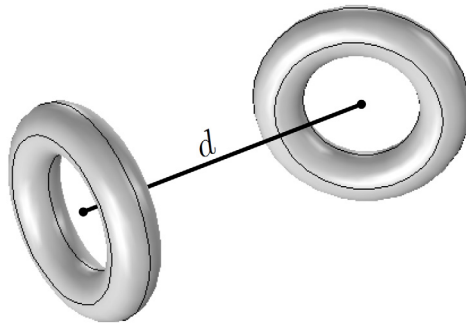
The **Fixed Joint** rigidly connects the two components and does not allow any relative motion between them. It does not add any degrees of freedom.



Fixed Joint Theory

Distance Joint

The **Distance Joint** maintains a specified distance between two points that are connected to a pair of attachments or rigid domains. As a default, the initial distance is maintained, so that the joint can be considered as a rigid link with ball joints at both ends. It is also possible to vary the distance so that the joint acts as a telescopic link.

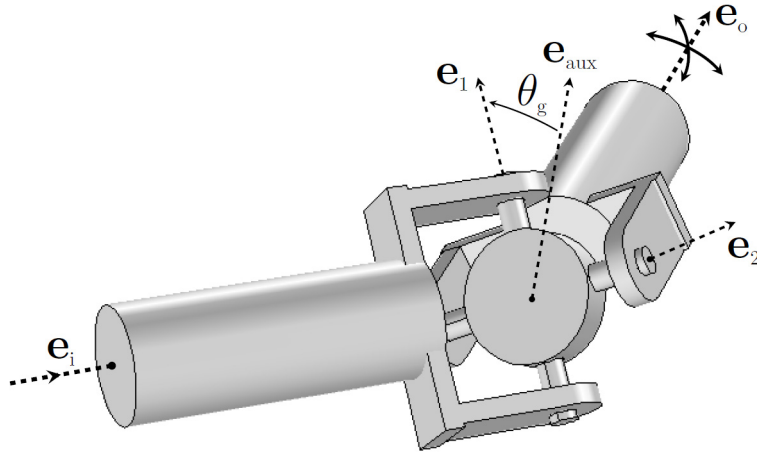


Fixed Joint Theory

Universal Joint

An **Universal Joint**, also known as Hooke's joint or Cardan joint, is commonly used to transmit rotational motion between shafts that have different orientations. Physically this joint is made up of a pair of hinge joints, oriented perpendicular to each other,

connected by a cross shaft or gimbal. This joint is a kind of abstract joint where the gimbal is not modeled and both the shafts are directly connected to each other. In general, this joint is used to transfer mechanical power from one shaft to another shaft. The added degrees of freedom in this joint are the relative rotations about the two legs of the gimbal that form the joint axis and second axis.



Universal Joint Theory

Attachments

An **Attachment** is a set of boundaries, edges, or points on a flexible or rigid component used to connect it to another flexible component or a rigid component through a joint or spring. An attachment can be defined on the boundaries of a solid element, on the edges of a shell element, and on the points of a beam element.

When an attachment is connected to a flexible body, you can use two different formulations: rigid and flexible.

In the rigid attachment formulation, all selected boundaries or edges behave as if they were connected to a common rigid body. This may cause an unrealistic stiffening and local stress concentrations.

In the flexible version, the boundaries are allowed to deform, and the rigid body constraints are enforced only in an average sense.

The attachment formulation is similar to that of a rigid connector. In the rigid case, the only degrees of freedom needed to represent this assembly are the ones needed to describe the movement of a rigid body. In 2D this is just two in-plane translations, and the rotation around the z -axis.

In 3D the situation is more complex. Six degrees of freedom are necessary, usually selected as three translations and three parameters for the rotation. For finite rotations any choice of three rotation parameters is singular at some specific set of angles. For this reason, a four-parameter quaternion representation is used for the rotations.

Some extra degrees of freedom are added for each attachment where the flexible formulation is used.

When an attachment is defined on a rigid component, it does not create any degrees of freedom of its own and directly picks the degrees of freedom of the rigid component.



The formulation of the attachment feature is same as the rigid connector formulation, which is discussed in [Rigid Connector Theory](#) in the *Structural Mechanics Module User's Guide*.



The links to sections described in external guides, such as the *Structural Mechanics Module User's Guide*, do not work in the PDF, only from the on line help in COMSOL Multiphysics.

Some useful information about the attachment feature:

- The attachment can be defined either on a flexible or on a rigid component. It is not possible to select the boundaries from both types of components in a single attachment. The reason is the different formulation of attachment for flexible or rigid components.
- The attachment center of rotation is the centroid of its selected boundaries, edges, or points. In a joint, it is possible to select the attachment center of rotation as the center of the joint.
- If the attachment is connected to a flexible domain, the forces and moments on an attachment with a boundary selection are computed by summing the reaction forces on the selected boundaries. There is an option to use these forces and moments to evaluate the joint forces.
- For an attachment in the Shell interface, the selection consists of edges. The forces and moments on an attachment are then computed by summing the reaction forces and reaction moments on the selected edges. There is an option to use these forces and moments to evaluate the joint forces.
- For an attachment in the Beam interface, the selection consists of points. The forces and moments on an attachment are then computed by summing the reaction forces and reaction moments on the selected points. There is an option to use these forces and moments to evaluate the joint forces.
- When an attachment is defined in the Shell or Beam interface, the normal force cannot be defined through **Attachment reaction forces** in the [Friction](#) subnode of a joint.
- In a joint you select attachments to establish its connection with flexible, or optionally, rigid bodies. This is accomplished by setting up a relation between the source attachment and destination attachment degrees of freedom. Thus the motion of an attachment is governed by the joints where this attachment is used.
- An attachment which is not referenced by any joint acts as an unconstrained rigid connector.



- [The Multibody Dynamics Interface](#)
 - [The Solid Mechanics Interface, and The Shell and Plate Interfaces, and The Beam Interface](#) in the *Structural Mechanics Module User's Guide*.
-

Cam-Follower

A cam is defined through a set of boundaries or edges that is being followed by a point called “follower”. The cam selection can be defined on rigid as well as on elastic bodies. Thus, cam boundaries or edges can undergo any kind of rigid body motion as well as deformation. When defining a cam selection, it should also be possible to use boundaries or edges that are defined outside the physics interface.

A follower is defined through a point which follows the cam boundaries or edges with certain optional offset. The follower point can be a part of rigid or elastic bodies.

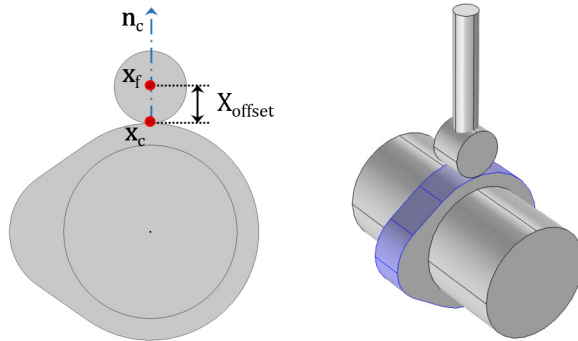


Figure 2-1: Example of a cam-follower connection.

The following settings help in avoiding convergence issues and getting more accurate results:

- The geometry shape order should be quadratic or higher. By default, it is the same as the shape order of the physics dependent variable. So choose quadratic as the discretization for the physics dependent variable.
- The finite element mesh should be sufficiently fine on the cam selection in order to get accurate spatial position, spatial normal, and spatial tangent of the closet point.
- The solver time steps should be sufficiently small such that the bodies do not rotate or move by a large amount in single time step.



Springs and Dampers

You can use a [Spring-Damper](#) to connect two points by an elastic spring, a viscous damper, or both. The two connection points are called the *source* and *destination* point, respectively.

- A source point can be placed on a flexible body, a rigid domain, an attachment, a fixed position, or a moving base.
- A destination point can be placed on a flexible body, an attachment, or a rigid domain.

When a flexible body is defined within the Multibody Dynamics interface, a connection point can be directly selected on the body. In the case when a flexible body is defined in another interface (Solid Mechanics, Shell, or Beam interface) an attachment is needed to define a connection point.

The two points can move relative to each other in an arbitrary way as long as they do not coincide. The spring and damping forces act along the line between the current positions of the two connection points.

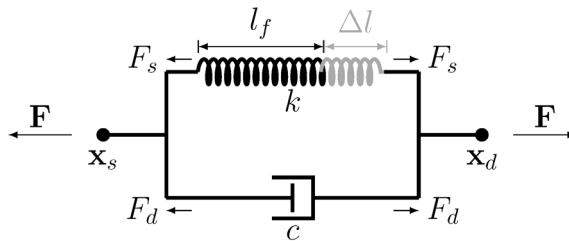


Figure 2-2: Conceptual sketch of a Spring-Damper.

You can specify that the Spring-Damper should be deactivated under certain conditions. It can for example be active only in tension, or break at a certain elongation.



Gears

About Gears

A gear is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the gear pair node while connecting to other gears. Similar to the rigid domain, the gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.



For details about the theory of a rigid body see [Rigid Domain Model](#) in the *Structural Mechanics Module User's Guide*.

The rest of this section has these topics:

- [Gear Types](#)
- [Connection with Other Gears](#)
- [Center of Rotation](#)
- [Gear Axis](#)
- [Mounting Methods](#)
- [Connecting to Other Physics Interfaces](#)

GEAR TYPES

There are six type of gears available:

- [Spur Gear](#)
- [Helical Gear](#)
- [Bevel Gear](#)
- [Worm Gear](#)
- [Spur Rack](#)
- [Helical Rack](#)

The bevel and worm gears are available only in 3D.

CONNECTION WITH OTHER GEARS

A gear can be connected to other gears through following nodes:

- [Gear Pair](#)
- [Worm and Wheel](#)
- [Rack and Pinion](#)

A single gear can be connected to one or more gears.

CENTER OF ROTATION

Gears are defined through rigid body degrees of freedom. These degrees of freedom are created at the center of rotation and the rotation is interpreted about this point. By default it is set to the center of mass, but there are other ways also to define it explicitly. This is the point where the forces and moment which act on the gear due to meshing with other gears are interpreted.

GEAR AXIS

The gear axis is the axis of rotation of the gear. It passes through the center of rotation. This axis is used to create the gear local coordinate system in the gear pair node. The gear rotation, a degree of freedom in the gear pair node, is also interpreted about this axis.

MOUNTING METHODS

The motion of a gear can be constrained directly using its subnode. Alternatively, it can be mounted on a shaft. There are several ways through which a gear can be mounted on a shaft. Some of the commonly used methods are as follows:

- Directly mounted on a flexible shaft
- Mounted on a rigid or flexible shaft through a rigid or elastic fixed joint
- Mounted on a rigid or flexible shaft through a rigid or elastic hinge joint
- Mounted on a ground (a fixed shaft) through a rigid or elastic hinge joint

CONNECTING TO OTHER PHYSICS INTERFACES

The multibody dynamics analysis of a geared system provides the contact force variation on the gear tooth and the vibration in various parts of the structure. These results can be further used for subsequent analysis.

- A stress analysis of a gear tooth or a full gear can be performed using the contact force obtained from the multibody dynamics analysis. This analysis can be performed within the Multibody Dynamics interface.
- A fatigue analysis of a gear tooth can be performed using the contact force variation over a mesh cycle or full revolution. This analysis can be performed using the Fatigue interface.
- A set of gears or a gear train operating at a high speed gives rise to high frequency vibration which results in noise. An acoustic analysis of a geared system can be performed using the normal acceleration of the vibrating surfaces. This can be

performed using the Pressure Acoustics, Transient or Pressure Acoustics, Frequency Domain interfaces.

About Gear Pairs

This section has these topics:

- [Gear Pair](#)
- [Worm and Wheel](#)
- [Rack and Pinion](#)
- [Gear Pair Compatibility Criteria](#)
- [Gear Elasticity](#)
- [Transmission Error](#)
- [Backlash](#)
- [Friction \(Gear\)](#)

GEAR PAIR

The gear pair node connects two spur gears, helical gears, or bevel gears in such a way that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of both the gears are independent of each other.

For a line contact model, one additional constraint is added to restrict the relative rotation about a line joining the two gear centers. When friction is included on a gear pair, frictional forces in the plane perpendicular to the line of action are added on both the gears.

The degrees of freedom at the gear pair are θ_{wh} and θ_{pn} . They are defined as the rotation of the wheel and pinion about the first axis of their respective local coordinate system.



[Theory for Gear Pairs](#)

WORM AND WHEEL

The worm and wheel node connects a worm gear with a spur or helical gear so that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of both the gears are independent of each other.

When friction is included on a worm and wheel node, frictional forces in the plane perpendicular to the line of action are added on both the gears.

The degrees of freedom at the worm and wheel are θ_{wm} and θ_{wh} . They are defined as the rotation of the worm and wheel about the first axis of their respective local coordinate system.



Theory for Worm and Wheel

RACK AND PINION

The rack and pinion node connects a spur or helical rack with a spur or helical gear in so that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of the rack and gear are independent of each other.

For a line contact model, one additional constraint is added to restrict the relative rotation about a line joining the rack and gear centers. If friction is included on a rack and pinion node, frictional forces in the plane perpendicular to the line of action are added on the rack and pinion.

The degrees of freedom at the rack and pinion are u_{rk} and θ_{pn} . They are defined as the displacement of rack along the third axis and the rotation of pinion about the first axis of their respective local coordinate system.



Theory for Rack and Pinion

GEAR PAIR COMPATIBILITY CRITERIA

In a gear pair node, you can select any two gears defined for the model. For the correct tooth meshing, a set of gears must however fulfill the following compatibility criteria:

Normal Module

The normal module of the both the gears must be same.

$$m_{n, wh} = m_{n, pn}$$

Pressure Angle

The pressure angle of the both the gears must be same.

$$\alpha_{wh} = \alpha_{pn}$$

Configuration

For parallel axis helical gears, the sum of the helix angles must be zero.

$$\beta_{wh} + \beta_{pn} = 0$$

For crossed axis helical gears, the sum of the helix angles must be nonzero.

$$\beta_{wh} + \beta_{pn} \neq 0$$

GEAR ELASTICITY

Gear elasticity can be added to gear pairs. In this node, elastic properties of the gear mesh such as mesh stiffness, mesh damping, and contact ratio can be specified. By default the gear mesh is assumed to be rigid.



[Theory for Gear Elasticity](#)

TRANSMISSION ERROR

A transmission error can be added to gear pairs. In this node, the static transmission error can be specified. It is added in the rotation constraint of the gear pair. By default there is no transmission error for a gear pair and it is assumed to be an ideal pair.

The static transmission error could be due to geometrical errors and geometrical modifications such as gear runouts, misalignments, tooth tip and root relief.



[Theory for Transmission Error](#)

BACKLASH

A backlash can be added to gear pairs. The transmission error due to backlash is added to the rotation constraint of the gear pair. By default there is no backlash for a gear pair.

The backlash on a gear pair plays an important role for unloaded gears. For loaded gears, it does not affect the dynamics of gears to a large extent.



[Theory for Backlash](#)

FRICTION (GEAR)

Friction can be added to gear pairs. When friction is included for a gear pair, frictional forces in the plane perpendicular to the line of action are added on both the gears.

By default there is no friction on [Gear Pair](#) and [Rack and Pinion](#). It is, however, included for [Worm and Wheel](#). The worm and wheel pair has always neither parallel nor intersecting configuration and in this configuration, there is always a slip even under perfect conditions.



Theory for Gear Friction

Gear Geometries

The Multibody Dynamics Module has a built-in geometry library for creating different types of gears. The gear geometries are available for 3D and 2D models. You can use them to build a gear tooth, a single gear, a pair of gears, or a gear train.

All the gear geometries are parameterized. You can vary input parameters to customize the gear tooth or gear blank shape.

To avoid building an invalid geometry, there are several checks on the input parameters. The set of input parameters must result in a valid gear geometry. In case of invalid set of parameters, an appropriate error message is displayed to adjust the specific parameter.

The geometries of the gear tooth cross sections are as follows:

- [Gear Tooth Geometry \(External: 2D\)](#)
- [Gear Tooth Geometry \(Internal: 2D\)](#)
- [Rack Tooth Geometry \(2D\)](#)

The gear geometries in 2D are as follows:

- [Spur Gear Geometry \(External: 2D\)](#)
- [Spur Gear Geometry \(Internal: 2D\)](#)
- [Spur Rack Geometry \(2D\)](#)

The gear geometries in 3D are as follows:

- [Spur Gear Geometry \(External: 3D\)](#)
- [Spur Gear Geometry \(Internal: 3D\)](#)
- [Helical Gear Geometry \(External: 3D\)](#)
- [Helical Gear Geometry \(Internal: 3D\)](#)
- [Bevel Gear Geometry \(3D\)](#)
- [Worm Gear Geometry \(3D\)](#)
- [Spur Rack Geometry \(3D\)](#)
- [Helical Rack Geometry \(3D\)](#)

Gear Tooth Geometry (External: 2D)

The part name for the tooth cross section of an external gear is `gear_tooth_2d`. This cross section of the gear tooth is in the normal plane of the gear.

The following sketch shows all the tooth parameters that can be varied to customize the tooth shape.

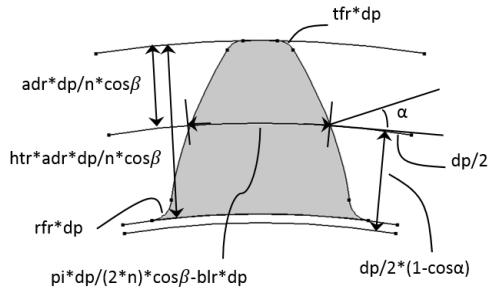


Figure 2-3: The sketch of the tooth cross section of an external gear.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Gear Tooth Geometry (Internal: 2D)

The part name for the tooth cross section of an internal gear is `internal_gear_tooth_2d`. This cross section of the gear tooth is in the normal plane of the gear.

The following sketch shows all the tooth parameters that can be varied to customize the tooth shape.

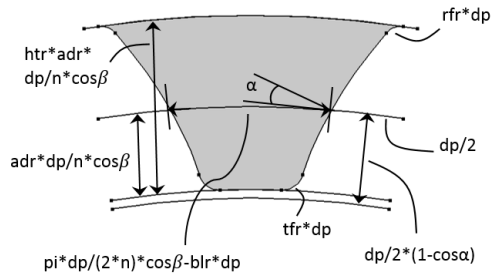


Figure 2-4: The sketch of the tooth cross-section of an internal gear.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Rack Tooth Geometry (2D)

The part name for the tooth cross section of a rack is rack_tooth_2d. This cross section of the gear tooth is in the normal plane of the gear.

The following sketch shows all the tooth parameters that can be varied to customize the tooth shape.

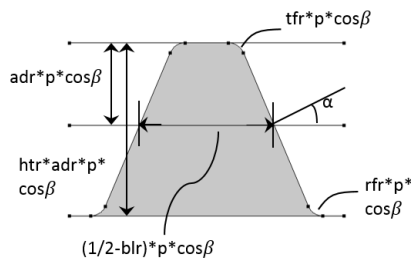


Figure 2-5: The sketch of the tooth cross section of a rack.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Spur Gear Geometry (External: 2D)

The part name for the external spur gear geometry in 2D is `spur_gear_2d`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

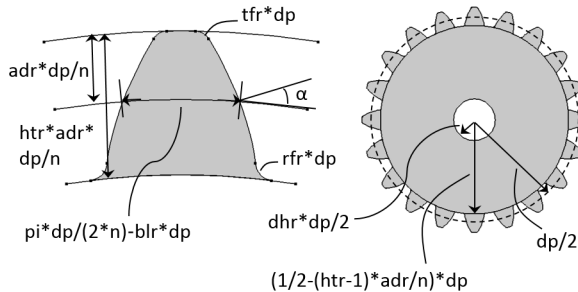


Figure 2-6: The sketch of the external spur gear geometry in 2D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root
- Hole at the gear center

Spur Gear Geometry (Internal: 2D)

The part name for the internal spur gear geometry in 2D is `internal_spur_gear_2d`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

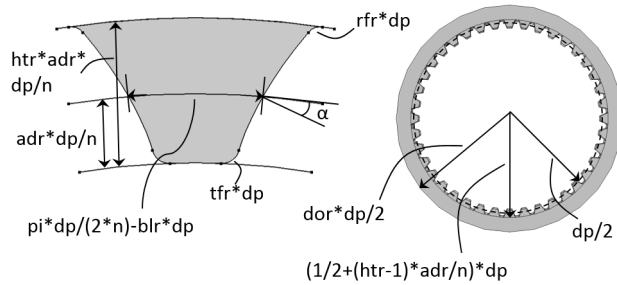


Figure 2-7: The sketch of the internal spur gear geometry in 2D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Spur Rack Geometry (2D)

The part name for the spur rack geometry in 2D is spur_rack_2d.

The following sketch shows all the tooth and rack parameters that can be varied to customize the available geometry.

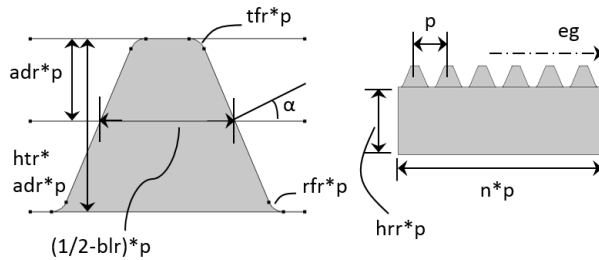


Figure 2-8: The sketch of the spur rack geometry in 2D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Spur Gear Geometry (External: 3D)

The part name for the external spur gear geometry in 3D is `spur_gear`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

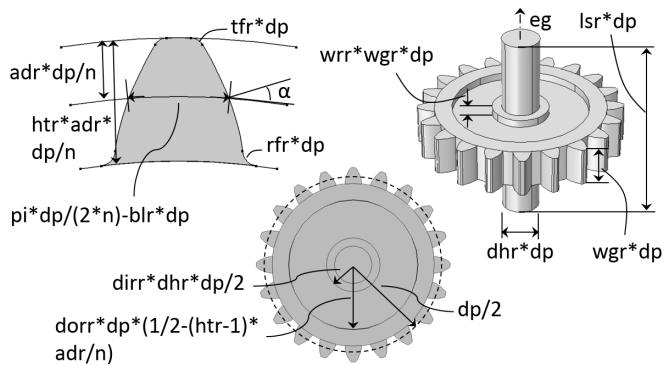


Figure 2-9: The sketch of the external spur gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root
- Addition of a shaft
- Hole at the gear center
- Material removal from the gear blank

Spur Gear Geometry (Internal: 3D)

The part name for the internal spur gear geometry in 3D is `internal1_spur_gear`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

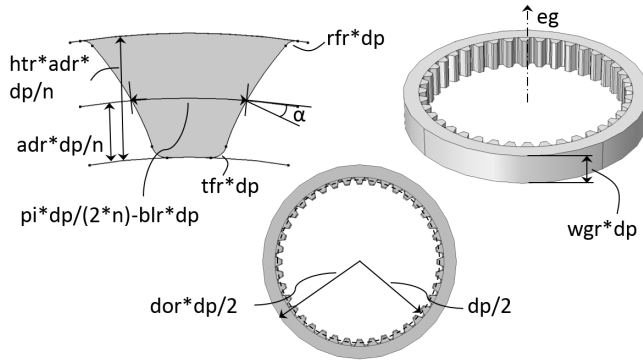


Figure 2-10: The sketch of the internal spur gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Helical Gear Geometry (External: 3D)

The part name for the external helical gear geometry in 3D is `helical_gear`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

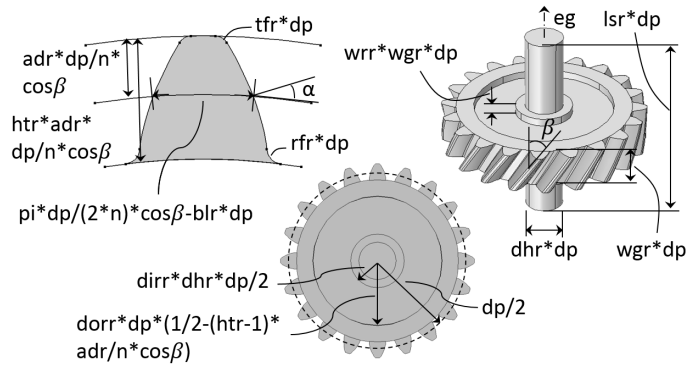


Figure 2-11: The sketch of the external helical gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root
- Addition of a shaft
- Hole at the gear center
- Material removal from the gear blank

Helical Gear Geometry (Internal: 3D)

The part name for the internal helical gear geometry in 3D is `internal_helical_gear`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

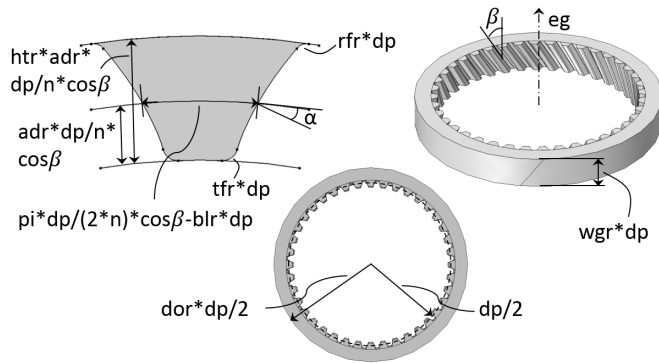


Figure 2-12: The sketch of the internal helical gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Bevel Gear Geometry (3D)

The bevel gear geometry is available only in 3D. The part name for this is `bevel_gear`.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

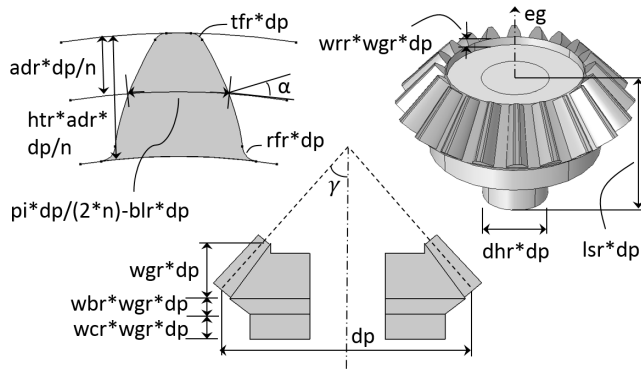


Figure 2-13: The sketch of the bevel gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root
- Addition of a shaft
- Hole at the gear center
- Material removal from the gear blank

Worm Gear Geometry (3D)

The worm gear geometry is available only in 3D. The part name for this is worm_gear.

The following sketch shows all the tooth and gear parameters that can be varied to customize the available geometry.

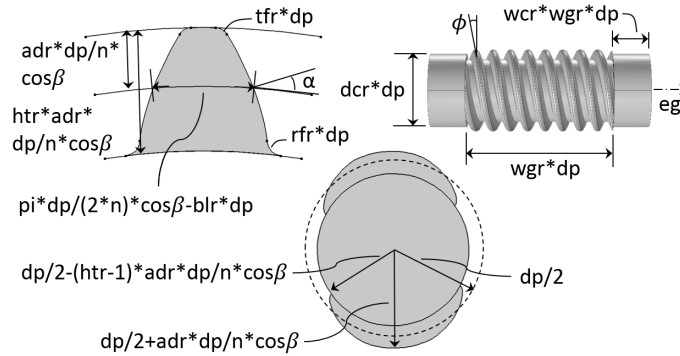


Figure 2-14: The sketch of the worm gear geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root
- Addition of collars

Spur Rack Geometry (3D)

The part name for the spur rack geometry in 3D is spur_rack.

The following sketch shows all the tooth and rack parameters that can be varied to customize the available geometry.

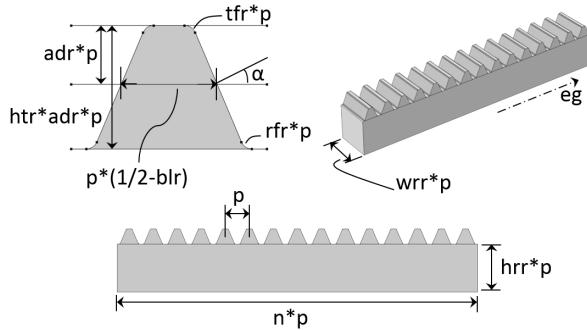


Figure 2-15: The sketch of the spur rack geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Helical Rack Geometry (3D)

The part name for the helical rack geometry in 3D is `helical_rack`.

The following sketch shows all the tooth and rack parameters that can be varied to customize the available geometry.

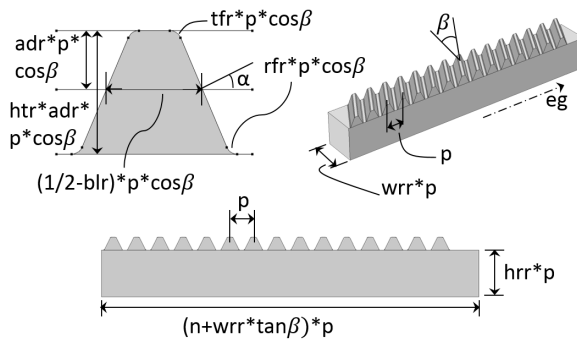


Figure 2-16: The sketch of the helical rack geometry in 3D.

OPTIONAL FEATURES

The following geometry operations are optional in this part:

- Fillet at the tooth tip
- Fillet at the tooth root

Bearings

Bearings are common in multibody systems, since they are necessary to support the relative rotation between two components. Depending on the relative rotational speed and magnitude of the load transfer between the components, different types of bearings are used, for example, rolling element bearings and fluid film bearings.

Roller bearings are often used for low speed applications where noise is not an important consideration. However, roller bearings perform poorly for impact loads. These bearings have a limited life, but can be easily replaced due to their low cost. In particular, misalignment can decrease their life expectancy significantly. Due to poor damping characteristics, these bearings are sometimes used with squeeze film dampers.

Fluid film bearings overcome all the above problems, but they are expensive to maintain. A complex lubrication system is needed to maintain the film thickness in the bearings and to avoid them from starving. Fluid film bearings also provide a good stability to unbalance excitation. However, fluid film bearings cannot be used for low speed applications. Other limitations of the fluid film bearings include requirement of a higher starting torque, high power loss, and temperature sensitivity.

One of the key criteria for selecting bearings is the combination of speed and load carrying capacity. The load carrying capacity of a fluid film bearing increases with the speed, while the opposite is true for rolling element bearings. Therefore, for a high load at high speeds, fluid film bearings are preferred.

Bearing Modeling

A number of approaches exist for modeling the bearings in an assembly. The simplest approach is to use equivalent stiffness and damping constants for the bearings in the system model. In this case, dynamic characteristics of the bearings are obtained either by experiments or by simulations. These characteristics then can directly be used in a multibody system model. In the Multibody Dynamics interface, you can use an elastic hinge joint and provide the bearing stiffness as an input on the **Joint Elasticity** node to model the bearings using this approach.

A more complex approach is to combine a bearing simulation together with the multibody simulation. It is possible to perform such simulations in Multibody Dynamics Module by using functionality from the Rotordynamics Module. The

following sections describe the modeling of radial roller bearings and hydrodynamic bearings together in a multibody dynamics context.

RADIAL ROLLER BEARING

A **Radial Roller Bearing** node can be added in the Multibody Dynamics interface when you have a license also for the Rotordynamics Module. You select boundaries in the multibody system where the inner race of the bearing is connected. The following bearing types are available:

- Deep Groove Ball Bearing
- Angular Contact Ball Bearing
- Self Aligning Ball Bearing
- Spherical Roller Bearing
- Cylindrical Roller Bearing
- Tapered Roller Bearing

Depending on the bearing type, you need to input the respective geometric and materials properties of the bearing components.

HYDRODYNAMIC BEARING

Modeling of hydrodynamic bearings in multibody requires a coupling to the Hydrodynamic Bearing interface in the Rotordynamics Module. When both a Multibody Dynamics interface and a Hydrodynamic Bearing interface are present in the model, a multiphysics coupling named **Solid-Bearing Coupling** can be added under the **Multiphysics** node in the Model Builder. Use this feature to combine the multibody and fluid film simulations. Add a **Hydrodynamic Journal Bearing** node in the Hydrodynamic Bearing interface to simulate different types of Journal Bearings. The following bearing types are available:

- Plain Journal Bearing
- Elliptic Journal Bearing
- Offset Halves Journal Bearing
- Multilobe Journal Bearing
- Tilted Pad Journal Bearing

You can also model a floating ring journal bearing using the **Floating Ring Bearing** node in the Hydrodynamic Bearing interface.

Initialization in Time-Dependent Studies

A time-dependent analysis must have appropriate initial conditions. If the analysis starts with nonzero initial conditions, you would in principle need to derive and input all components of the initial values for all parts of the structure. In a complex assembly, this may require some effort.

In a time-dependent analysis, there is a step where consistent initial values can be computed. This step ensures, by modifying the given initial conditions, that the initial values actually used are matching everywhere. The new initial values can be seen as a type of average of the ones you entered, if they were not consistent.

In a multibody dynamics analysis, it is common that you know the initial conditions for a certain component, and want those initial values to be exactly fulfilled. The initial values for other parts should then be computed so that they match the ones prescribed. The initial value settings in the Multibody Dynamics Module gives you an option for doing so. The computation of consistent initial values is still performed, but the values you have marked by setting the control **Consistent initialization to Force initial values** are respected.

If this option is used, the degrees of freedom for which **Force initial values** is selected are treated as having prescribed displacements or rotations during the initialization step.

Even if the initial values of certain components are forced in a multibody system containing flexible bodies, it is possible that the system is not initialized with the known (forced) initial values because of the deformation of the elastic components during the consistent initialization step. To initialize the system with the known initial values like a rigid body system, elastic bodies can be assumed rigid during the consistent initialization step. You can do that by adding an **Initially rigid** node, where you select the flexible domains which should be treated as rigid during the initialization.

Geometric Nonlinearity

In most cases, the parts in a multibody analysis will translate and rotate significantly during the analysis. This means that the analysis must be considered as geometrically nonlinear.

You enable a geometrically nonlinear analysis for a certain study step by selecting the **Include geometric nonlinearity** check box in the **Study Settings** section of a study step. If any active feature in the model requires the analysis to be geometrically nonlinear, the **Include geometric nonlinearity** check box is selected automatically, and it cannot be cleared. The physics features within the Multibody Dynamics interface which force this behavior are:

- Any **Joint**.
- When a nonzero rotation is prescribed in a **Prescribed Displacement/Rotation** subnode of **Rigid Domain**.
- When a nonzero rotation is prescribed for a **Rigid Connector**.
- When a **Contact** node is used.



- [Modeling Geometric Nonlinearity](#)
 - [Joints](#)
 - [Contact Modeling](#)
-

Multiphysics Couplings

It is possible to couple the Multibody Dynamics interface to several other physics interfaces through predefined multiphysics couplings. This section describes the available couplings.

When you add a physics interface for which a predefined coupling is available to the component, a **Multiphysics Couplings** node appears in the model tree. Under the **Multiphysics Couplings** node, you can add the required couplings.

TABLE 2-3: AVAILABLE MULTIPHYSICS COUPLINGS FOR THE MULTIBODY DYNAMICS INTERFACE

COUPLING	INTERFACES	COMMENT
Thermal Expansion	Heat Transfer Heat Transfer in Solids Heat Transfer in Porous Media Bioheat Transfer	One-way domain coupling
Temperature Coupling	Heat Transfer Heat Transfer in Solids Heat Transfer in Porous Media Bioheat Transfer	Used together with Thermal Expansion
Acoustic-Structure Boundary	Pressure Acoustics, Frequency Domain Pressure Acoustics, Time Domain	Two-way boundary coupling
Thermoviscous Acoustics-Structure Boundary	Thermoviscous Acoustics, Frequency Domain	Two-way boundary coupling
Fluid-Structure Interaction	Any single phase flow interface	One-way or two-way boundary coupling
Fluid-Structure Interaction Pair	Any single phase flow interface	Two-way boundary coupling
Solid-Bearing Coupling	Hydrodynamic Bearing	Two-way boundary coupling

There are two multiphysics interfaces which are specific to the Multibody Dynamics Module:


- [The Fluid-Multibody Interaction Interface](#)
- [The Fluid-Multibody Interaction, Assembly Interface](#)



The other couplings are analogous to those available for the Solid Mechanics interface. See also:



- [Defining Multiphysics Models](#) in the *Structural Mechanics Module User's Guide*
- [Thermally Coupled Problems](#) in the *Structural Mechanics Module User's Guide*
- [Fluid-Structure Interaction](#) in the *Structural Mechanics Module User's Guide*
- More details about Solid-Bearing Coupling in the *Rotordynamics Module User's Guide*

The Fluid-Multibody Interaction Interface

Using the **Fluid-Multibody Interaction** () interface, you can model phenomena where a fluid and a rigid or deformable solid affect each other. The interface includes a predefined condition for the interaction at the fluid-solid boundaries. [Arbitrary Lagrangian-Eulerian Formulation \(ALE\)](#) is used for incorporating the geometrical changes of the fluid domain.

Use this interface if the geometry sequence is set up to form a union. In multibody modeling, it is more common that the assembly mode is used, in which case you should use the [The Fluid-Multibody Interaction, Assembly Interface](#).

Using a stationary or a time-dependent study, the **Fluid-Solid Interaction** interface models two-way coupling between solids and fluids. There are, however, special study steps available in order to model one-way coupled fluid-structure interaction.

You add a predefined **Fluid-Solid Multibody** interface from the **Fluid-Structure Interaction** () group in the **Fluid Flow** branch () of the **Model Wizard** or **Add Physics** windows. **Laminar Flow** and **Multibody Dynamics** interfaces are then added to the Model Builder.

At the same time, a **Moving Mesh** node with a **Deforming Domain** subnode is created under **Definitions**. For more information, see [Deforming Fluid Domains](#).

In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Fluid-Structure Interaction**.

On the Constituent Physics Interfaces

- The Laminar Flow interface solves the Navier-Stokes equations for the conservation of momentum and mass, to compute the fluid velocity and pressure. The interface supports incompressible flows and compressible flows at low Mach numbers (typically less than 0.3). When the CFD Module is available, turbulent flow is supported through the inclusion of a number of turbulence models.



If you also have the CFD Module, you can add a **Stationary, One-Way Coupled with Initialization** or **Transient, One-Way Coupled with Initialization** study for turbulence models requiring the wall distance. See [Stationary, One-Way Coupled with Initialization](#) and [Time Dependent, One-Way Coupled with Initialization](#) in the *COMSOL Multiphysics Reference Manual* for more information.

- The Multibody Dynamics interface is intended for analysis of mechanical assemblies connected by various types of joints and spring-dampers. There are many types of joints, such as hinges or ball joints, that can be used based on the type of connection required between the two components. The joints can be given properties such as spring constants, damping, friction, and limits on movement. The joint can also be made elastic to model the bushing between the two components. Computed results are displacements, velocities, accelerations, joint forces, and stresses in flexible parts.



In 2D, plane stress or plane strain assumptions can be selected in the Solid Mechanics interface. Only the plane strain (default) should be used together with the FSI coupling.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, all specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single Laminar Flow and Multibody Dynamics interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose from the available coupling features, but the modified settings are not included.

Another difference is that the Laminar Flow interface added by the predefined coupling employs pseudo-time stepping per default for stationary models, while the standalone version uses an automatic Newton damping. The latter can sometimes give shorter solution times, while the pseudo-time stepping often gives a more robust solution procedure. You can disable the pseudo-time stepping in **Advanced Settings** section for the fluid interface.



Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES



Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Features

The [Fluid-Structure Interaction](#) multiphysics coupling is used to couple the interfaces.

Physics Interface Features


Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.



- The available physics features for [The Single-Phase Flow, Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *COMSOL Multiphysics Reference Manual*.
- The available physics features for [The Multibody Dynamics Interface](#) are listed in the [Domain, Boundary, Edge, Point, and Pair Nodes for the Multibody Dynamics Interface](#) section.

The Fluid-Multibody Interaction, Assembly Interface

Using the **Fluid-Multibody Interaction, Assembly** () interface, you can model phenomena where a fluid and a rigid or deformable solid affect each other. The interface includes a predefined condition for the interaction at the fluid-solid boundaries. [Arbitrary Lagrangian-Eulerian Formulation \(ALE\)](#) is used for incorporating the geometrical changes of the fluid domain.

Use this interface if the geometry sequence is set up to form an assembly.

Using a stationary or a time-dependent study, the **Fluid-Solid Interaction** interface models two-way coupling between solids and fluids. There are, however, special study steps available in order to model one-way coupled fluid-structure interaction.

You add a predefined **Fluid-Multibody, Assembly** interface from the **Fluid-Structure Interaction** () group in the **Fluid Flow** branch () of the **Model Wizard** or **Add Physics** windows. **Laminar Flow** and **Multibody Dynamics** interfaces are then added to the Model Builder.

At the same time, a **Moving Mesh** node with a **Deforming Domain** subnode is created under **Definitions**. For more information, see [Deforming Fluid Domains](#).

In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Fluid-Structure Interaction, Pair**.

On the Constituent Physics Interfaces

- The Laminar Flow interface solves the Navier-Stokes equations for the conservation of momentum and mass, to compute the fluid velocity and pressure. The interface supports incompressible flows and compressible flows at low Mach numbers (typically less than 0.3). When the CFD Module is available, turbulent flow is supported through the inclusion of a number of turbulence models.



If you also have the CFD Module, you can add a **Stationary, One-Way Coupled with Initialization** or **Transient, One-Way Coupled with Initialization** study for turbulence models requiring the wall distance. See [Stationary, One-Way Coupled with Initialization](#) and [Time Dependent, One-Way Coupled with Initialization](#) in the *COMSOL Multiphysics Reference Manual* for more information.

- The Multibody Dynamics interface is intended for analysis of mechanical assemblies connected by various types of joints and spring-dampers. There are many types of

joints, such as hinges or ball joints, that can be used based on the type of connection required between the two components. The joints can be given properties such as spring constants, damping, friction, and limits on movement. The joint can also be made elastic to model the bushing between the two components. Computed results are displacements, velocities, accelerations, joint forces, and stresses in flexible parts.



In 2D, plane stress or plane strain assumptions can be selected in the Solid Mechanics interface. Only the plane strain (default) should be used together with the FSI coupling.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, all specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single Laminar Flow and Multibody Dynamics interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose from the available coupling features, but the modified settings are not included.

Another difference is that the Laminar Flow interface added by the predefined coupling employs pseudo-time stepping per default for stationary models, while the standalone version uses an automatic Newton damping. The latter can sometimes give shorter solution times, while the pseudo-time stepping often gives a more robust solution procedure. You can disable the pseudo-time stepping in **Advanced Settings** section for the fluid interface.



Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES



Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Features

The [Fluid-Structure Interaction, Pair](#) multiphysics coupling is used to couple the interfaces.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for [The Single-Phase Flow, Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *COMSOL Multiphysics Reference Manual*.
- The available physics features for [The Multibody Dynamics Interface](#) are listed in the [Domain, Boundary, Edge, Point, and Pair Nodes for the Multibody Dynamics Interface](#) section.

Multibody Dynamics Theory

This chapter introduces you to the theory which is specific for the Multibody Dynamics interface features.

In this chapter:

- [Joint Theory](#)
- [Cam-Follower Theory](#)
- [Spring-Damper Theory](#)
- [Gear Theory](#)
- [Gear Pair Theory](#)

Joint Theory

In this section:

- Representation of Rotations
- Prismatic Joint Theory
- Hinge Joint Theory
- Cylindrical Joint Theory
- Screw Joint Theory
- Planar Joint Theory
- Ball Joint Theory
- Slot Joint Theory
- Reduced Slot Joint Theory
- Fixed Joint Theory
- Distance Joint Theory
- Universal Joint Theory
- Theory for Joint Constraints
- Theory for Joint Locking
- Theory for Springs and Dampers on Joints
- Theory for Prescribed Motion on Joints
- Theory for Applied Forces and Moments on Joints
- Theory for Friction in Joints
- Theory for Joint Forces and Moments Computation

Representation of Rotations

In 2D models, the rotation can only occur around the z -axis. It is represented by a scalar angle, θ .

In 3D the situation is more complex. There are three rotational degrees of freedom. For finite rotations, however, any choice of three rotation parameters is singular at some specific set of angles. For this reason, a four-parameter *quaternion* representation is used for the rotations in the joints. The quaternion parameters are called a , b , c , and d , respectively. These four parameters are not independent, so an extra equation stating that the following relation is added:

$$a^2 + b^2 + c^2 + d^2 = 1$$

The four quaternion parameters are often represented as one scalar and one vector,

$$\mathbf{q} = \begin{bmatrix} a \\ \mathbf{b} \end{bmatrix}$$

where

$$\mathbf{b} = \begin{bmatrix} b \\ c \\ d \end{bmatrix}$$

The connection between the quaternion parameters and a rotation matrix R is

$$R = \begin{bmatrix} a^2 + b^2 - c^2 - d^2 & 2bc - 2ad & 2ac + 2bd \\ 2ad + 2bc & a^2 - b^2 + c^2 - d^2 & 2cd - 2ab \\ 2bd - 2ac & 2ab + 2cd & a^2 - b^2 - c^2 + d^2 \end{bmatrix}$$

The rotation can be also represented as a rotation vector with direction \mathbf{e}_Θ and a magnitude, Θ .

$$\Theta = \Theta \mathbf{e}_\Theta$$

The magnitude of the rotation is

$$\Theta = 2 \arccos(a)$$

and the direction is

$$\mathbf{e}_\Theta = \frac{\mathbf{b}}{\|\mathbf{b}\|}$$

The parameter a can thus be considered as measuring the rotation, while b , c , and d can be interpreted as the orientation of the rotation vector. For small rotations, this relation simplifies to

$$\Theta = 2 \begin{bmatrix} b \\ c \\ d \end{bmatrix}$$

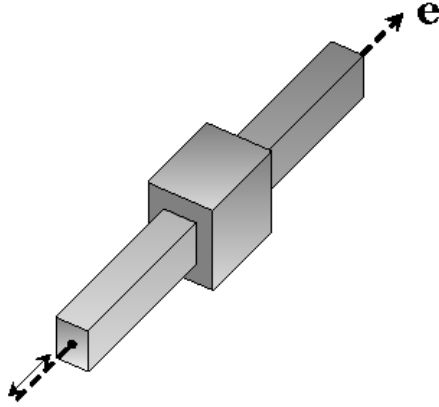
Successive rotations are represented by a quaternion multiplication

$$\begin{bmatrix} a_{tot} \\ \mathbf{b}_{tot} \end{bmatrix} = \begin{bmatrix} a_1 \\ \mathbf{b}_1 \end{bmatrix} \otimes \begin{bmatrix} a_2 \\ \mathbf{b}_2 \end{bmatrix} = \begin{bmatrix} a_1 a_2 - \mathbf{b}_1 \cdot \mathbf{b}_2 \\ a_1 \mathbf{b}_2 + a_2 \mathbf{b}_1 + \mathbf{b}_1 \times \mathbf{b}_2 \end{bmatrix}$$

Just as finite rotations, quaternion multiplications are not commutative.

Prismatic Joint Theory

The **Prismatic Joint**, also known as a translational or sliding joint, has one translational degree of freedom between the two connected components. The components are free to translate, or slide, relative to each other along the joint axis.



JOINT AXIS AND ITS LOCAL COORDINATE SYSTEM

The local coordinate system of a joint, which can be seen as rigidly connected to the source attachment, consists of three axes: joint axis (\mathbf{e}_1), second axis (\mathbf{e}_2), and third axis (\mathbf{e}_3). You specify the initial direction joint axis (\mathbf{e}_{10}), using one of the following methods:

- **Method 1:** Give the components explicitly in a selected coordinate system.
- **Method 2:** Select one of the axes of a selected coordinate system.
- **Method 3:** Select an edge that is parallel to the joint axis.



For all methods, the axis is normalized to give a unit vector.

In Method 3, the direction that has a positive projection on the global x -axis is selected. If the projection on the x -axis is very small (the edge is almost perpendicular to the x -axis), then the y -axis is used in the projection criterion. If the edge is perpendicular also to the y -axis, then the z -axis is used as the joint axis. You can also choose to reverse the direction of initial joint axis.

If the directions of the second and third axis are important (for example, when interpreting joint force results in the joint local coordinate system, or when there is elasticity in the joint), then Method 2 is preferred because this gives full control over all three axes. When an axis in the selected coordinate system is chosen to specify the joint axis, the other two axes are determined through cyclic permutation.

For Method 2 and Method 3, the initial second axis and initial third axis are computed internally as described below.

The initial second axis (\mathbf{e}_{20}) (perpendicular to the initial joint axis) is computed by making an *auxiliary axis* (\mathbf{e}_{aux}) (an axis not parallel to the initial joint axis) and then taking a cross product of the initial joint axis and this auxiliary axis:

$$\mathbf{e}_{20} = \mathbf{e}_{10} \times \mathbf{e}_{\text{aux}}$$

The initial third axis (\mathbf{e}_{30}) is computed by taking a cross product of the initial joint axis and the initial second axis:

$$\mathbf{e}_{30} = \mathbf{e}_{10} \times \mathbf{e}_{20}$$

Finally, the rotated axes in the deformed configuration are formed by rotating the individual axes using the rotation matrix of the source attachment (R_{src}):

$$\mathbf{e}_1 = R_{\text{src}} \cdot \mathbf{e}_{10}$$

$$\mathbf{e}_2 = R_{\text{src}} \cdot \mathbf{e}_{20}$$

$$\mathbf{e}_3 = R_{\text{src}} \cdot \mathbf{e}_{30}$$

For 2D models, the generation of the coordinate system is significantly simplified. The initial third axis (\mathbf{e}_{30}) is taken as the out-of-plane axis (\mathbf{e}_z). The initial second axis (\mathbf{e}_{20}) is orthogonal to (\mathbf{e}_{10}) and (\mathbf{e}_{30}):

$$\mathbf{e}_{20} = \mathbf{e}_z \times \mathbf{e}_{10}$$

$$\mathbf{e}_{30} = \mathbf{e}_z$$

PRISMATIC JOINT FORMULATION

For a prismatic joint, the destination attachment is free to translate relative to the source attachment along the joint axis. The added degree of freedom is thus the relative displacement along the joint axis (u).

The translational motion of the source and destination attachments at the center of the joint can be computed as:

$$\begin{aligned}\mathbf{u}_{c,src} &= (\mathbf{R}_{src} - \mathbf{I}) \cdot (\mathbf{X}_c - \mathbf{X}_{c,src}) + \mathbf{u}_{src} \\ \mathbf{u}_{c,dst} &= (\mathbf{R}_{dst} - \mathbf{I}) \cdot (\mathbf{X}_c - \mathbf{X}_{c,dst}) + \mathbf{u}_{dst}\end{aligned}$$

where, for the source and destination attachments, respectively:

- $\mathbf{u}_{c,src}$ and $\mathbf{u}_{c,dst}$ are the displacement vectors for the attachments at the center of joint.
- \mathbf{R}_{src} and \mathbf{R}_{dst} are the rotation matrices describing the rotation of each attachment.
- $\mathbf{X}_{c,src}$ and $\mathbf{X}_{c,dst}$ are the positions of the centroids of the attachments.
- \mathbf{u}_{src} and \mathbf{u}_{dst} are the displacements at the centroids of the attachments.
- \mathbf{X}_c is the joint center.

To formulate this kind of connection for a rigid joint, the motion of the destination attachment is prescribed in terms of the motion of the source attachment:

$$\begin{aligned}\mathbf{u}_{c,dst} &= \mathbf{u}_{c,src} + \mathbf{u}_r \\ \mathbf{b}_{dst} &= \mathbf{b}_{src}\end{aligned}$$

The displacement vector (\mathbf{u}) and quaternion vector (\mathbf{b}) for the source and destination attachments are defined as:

$$\mathbf{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}, \quad \mathbf{b} = \begin{bmatrix} b \\ c \\ d \end{bmatrix}$$

The relative displacement vector in the global spatial coordinate system (\mathbf{u}_r) is defined as:

$$\mathbf{u}_r = u \mathbf{e}_1$$

For 2D, the motion of the destination attachment is prescribed in terms of the motion of source attachment:

$$\begin{aligned}\mathbf{u}_{c,dst} &= \mathbf{u}_{c,src} + \mathbf{u}_r \\ \phi_{dst} &= \phi_{src}\end{aligned}$$

where ϕ is the rotation about z -axis (out-of-plane axis).

Joint Elasticity in Prismatic Joint

In the case that the joint is elastic, the elastic displacements and rotations are added to the source motion, so that the displacements and rotations at the center of joint is no longer equal for the source and destination attachments.

$$\begin{aligned}\mathbf{u}_{c, \text{dst}} &= \mathbf{u}_{c, \text{src}} + \mathbf{u}_{e1} + \mathbf{u}_r \\ \mathbf{b}_{\text{dst}} &= \tilde{\mathbf{b}}\end{aligned}$$

$$\begin{bmatrix} \tilde{a} \\ \tilde{\mathbf{b}} \end{bmatrix} = \begin{bmatrix} a_{e1} \\ \mathbf{b}_{e1} \end{bmatrix} \otimes \begin{bmatrix} a_{\text{src}} \\ \mathbf{b}_{\text{src}} \end{bmatrix}$$

The elastic translation and rotation vectors are expressed in terms of the joint axes as

$$\begin{aligned}\mathbf{u}_{e1} &= u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3 \\ \theta_{e1} &= \theta_1 \mathbf{e}_1 + \theta_2 \mathbf{e}_2 + \theta_3 \mathbf{e}_3\end{aligned}$$

Here, $u_2, u_3, \theta_1, \theta_2,$ and θ_3 are the default elastic degrees of freedom available for this joint. Optionally, only a subset of them can be chosen to be elastic.

The quaternion representing the elastic rotation is

$$\begin{bmatrix} a_{e1} \\ \mathbf{b}_{e1} \end{bmatrix}$$

and can be computed from the elastic rotation vector θ_{e1} .

The spring and damping forces and moments are proportional to the elastic displacements and velocities.

$$\begin{bmatrix} \mathbf{F}_{s, e1} \\ \mathbf{M}_{s, e1} \end{bmatrix} = \begin{bmatrix} \mathbf{k}_u & \mathbf{k}_{u\theta} \\ \mathbf{k}_{\theta u} & \mathbf{k}_\theta \end{bmatrix} \begin{bmatrix} \mathbf{u}_{e1} \\ \theta_{e1} \end{bmatrix}$$

$$\begin{bmatrix} \mathbf{F}_{d, e1} \\ \mathbf{M}_{d, e1} \end{bmatrix} = \begin{bmatrix} \mathbf{c}_u & \mathbf{c}_{u\theta} \\ \mathbf{c}_{\theta u} & \mathbf{c}_\theta \end{bmatrix} \begin{bmatrix} \dot{\mathbf{u}}_{e1} \\ \dot{\theta}_{e1} \end{bmatrix}$$

The spring and damping matrices are subdivided into four parts depending on whether they relate to translation or rotation. The matrices which couple translation and

rotations are optional in the input data. The total elastic joint forces and moments can be written using the spring and damping forces and moments as

$$\mathbf{F}_{el} = -\mathbf{F}_{s,el} - \mathbf{F}_{d,el}$$

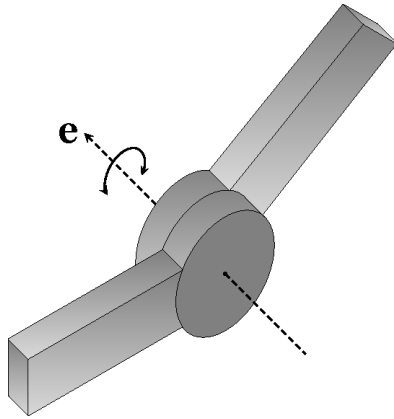
$$\mathbf{M}_{el} = -\mathbf{M}_{s,el} - \mathbf{M}_{d,el}$$

The contribution to the virtual work is

$$\delta W = \mathbf{F}_{el} \cdot \delta \mathbf{u}_{el} + \mathbf{M}_{el} \cdot \delta \theta_{el}$$

Hinge Joint Theory

The [Hinge Joint](#), also known as a revolute joint, has one rotational degree of freedom between the two components. The two components are free to rotate relative to each other about the axis of the joint.



The following is an addition to the [Prismatic Joint Theory](#), some of which is also applicable for the hinge joint.

HINGE JOINT AXIS AND ITS LOCAL COORDINATE SYSTEM

In 2D, the initial joint axes (\mathbf{e}_{10} , \mathbf{e}_{20} , and \mathbf{e}_{30}) are always global spatial axes with the initial joint axis (\mathbf{e}_{10}) being the out-of-plane axis (\mathbf{e}_z):

$$\mathbf{e}_{10} = \mathbf{e}_z$$

$$\mathbf{e}_{20} = \mathbf{e}_x$$

$$\mathbf{e}_{30} = \mathbf{e}_y$$

HINGE JOINT FORMULATION

For a hinge joint, the destination attachment is free to rotate relative to the source attachment about the joint axis. The relative rotation about the joint axis (θ) is the degree of freedom.

To formulate this kind of connection, the motion of the destination attachment is prescribed in terms of the motion of the source attachment with the following relations:

$$\begin{aligned}\mathbf{b}_{\text{dst}} &= \tilde{\mathbf{b}} \\ \mathbf{u}_{\text{c, dst}} &= \mathbf{u}_{\text{c, src}}\end{aligned}$$

$$\begin{bmatrix} \tilde{a} \\ \tilde{\mathbf{b}} \end{bmatrix} = \begin{bmatrix} a_r \\ \mathbf{b}_r \end{bmatrix} \otimes \begin{bmatrix} a_{\text{src}} \\ \mathbf{b}_{\text{src}} \end{bmatrix}$$

The relative quaternion in the global spatial coordinate system (a_r, \mathbf{b}_r) is defined as:

$$a_r = \cos(\theta/2)$$

$$\mathbf{b}_r = \sin(\theta/2)\mathbf{e}_1$$

In 2D, the motion of the destination attachment is prescribed in terms of the motion of the source attachment by:

$$\phi_{\text{dst}} = \phi_{\text{src}} + \theta$$

$$\mathbf{u}_{\text{c, dst}} = \mathbf{u}_{\text{c, src}}$$

Joint Elasticity in Hinge Joint

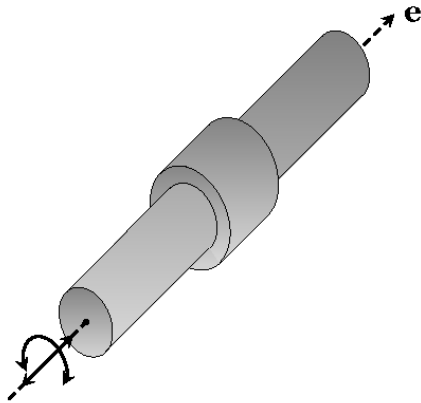
The elastic degrees of freedom are written as

$$\mathbf{u}_{\text{e1}} = u_1\mathbf{e}_1 + u_2\mathbf{e}_2 + u_3\mathbf{e}_3$$

$$\theta_{\text{e1}} = \theta_2\mathbf{e}_2 + \theta_3\mathbf{e}_3$$

Cylindrical Joint Theory

The [Cylindrical Joint](#) has one translational and one rotational degree of freedom between the two connected components. The components are free to slide and rotate relative to each other along the axis of joint. A cylindrical joint can be thought of as a combination of a prismatic joint and a hinge joint.



The following is an addition to the [Prismatic Joint Theory](#) and the [Hinge Joint Theory](#), which are also applicable for the cylindrical joint.

CYLINDRICAL JOINT FORMULATION

For a cylindrical joint, the destination attachment is free to translate and rotate relative to the source attachment along the axis of joint. Thus the degrees of freedom in this joint are the relative displacement (u) along the joint axis and the relative rotation (θ) about the joint axis.

To formulate this kind of connection, the motion of the destination attachment is prescribed in terms of the motion of the source attachment as:

$$\mathbf{b}_{\text{dst}} = \tilde{\mathbf{b}}$$
$$\mathbf{u}_{\text{c, dst}} = \mathbf{u}_{\text{c, src}} + \mathbf{u}_{\text{r}}$$

$$\begin{bmatrix} \tilde{a} \\ \tilde{\mathbf{b}} \end{bmatrix} = \begin{bmatrix} a_r \\ \mathbf{b}_r \end{bmatrix} \otimes \begin{bmatrix} a_{\text{src}} \\ \mathbf{b}_{\text{src}} \end{bmatrix}$$

The relative quaternion (a_r, \mathbf{b}_r) and the relative displacement vector (\mathbf{u}_r) in the global spatial coordinate system are defined as:

$$\begin{aligned} a_r &= \cos(\theta/2) \\ \mathbf{b}_r &= \sin(\theta/2)\mathbf{e}_1 \\ \mathbf{u}_r &= u\mathbf{e}_1 \end{aligned}$$

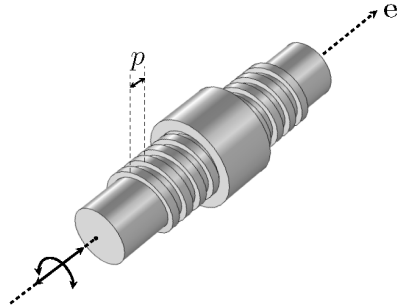
Joint Elasticity in Cylindrical Joint

The elastic degrees of freedom are written as

$$\begin{aligned} \mathbf{u}_{e1} &= u_2\mathbf{e}_2 + u_3\mathbf{e}_3 \\ \theta_{e1} &= \theta_2\mathbf{e}_2 + \theta_3\mathbf{e}_3 \end{aligned}$$

Screw Joint Theory

The **Screw Joint** is similar to the cylindrical joint except that the relative displacement and the relative rotation, instead of being independent, are linearly related to each other. Hence, this joint effectively has only one degree of freedom.



The following is an addition to the [Cylindrical Joint Theory](#), which is also applicable for the screw joint.

SCREW JOINT FORMULATION

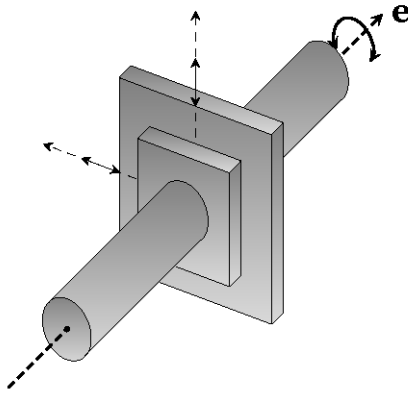
For the screw joint, an extra constraint is added to enforce the relation between the relative displacement (u) and the relative rotation (θ):

$$u = n \cdot p \cdot \frac{\theta}{2\pi}$$

Here n and p are the number of starts and the pitch of the screw thread, respectively.

Planar Joint Theory

The **Planar Joint** has two translational and one rotational degrees of freedom between the two components. Both components are free to translate relative to each other in a plane perpendicular to the axis of joint, and they are also free to rotate about this axis.



The following is an addition to the [Cylindrical Joint Theory](#), which is also applicable to the planar joint.

PLANAR JOINT FORMULATION

For a planar joint, the destination boundaries are free to translate relative to the source boundaries in a plane perpendicular to the joint axis and free to rotate relative to the source boundaries about this axis. The degrees of freedom are two relative displacements along the second and third axes of the joint (u_2, u_3) and one relative rotation about the joint axis (θ).

The joint formulation is similar to the cylindrical joint; the only difference is in the definition of the relative displacement vector (\mathbf{u}_r):

$$\mathbf{u}_r = u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3$$

Joint Elasticity in Planar Joint

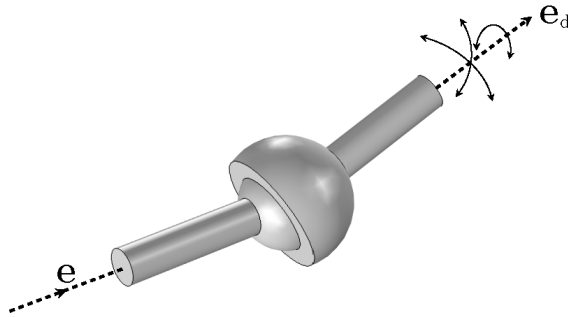
The elastic degrees of freedom are written as

$$\mathbf{u}_{e1} = u_1 \mathbf{e}_1$$

$$\theta_{e1} = \theta_2 \mathbf{e}_2 + \theta_3 \mathbf{e}_3$$

Ball Joint Theory

The **Ball Joint**, also known as a spherical joint, has three rotational degrees of freedom between the two attached components. The two components are free to rotate relative to each other along all three axes.



The following is an addition to the [Hinge Joint Theory](#), some of which is also applicable to the ball joint.

BALL JOINT AXIS AND ITS LOCAL COORDINATE SYSTEM

Unlike other joints, the ball joint has no particular axis about which the relative degrees of freedom between the two components are possible. This joint has rotational symmetry, and hence, relative rotation is allowed about any axis. However, application of constraints, springs, dampers, and so forth needs the source (\mathbf{e}_1) and destination (\mathbf{e}_{d1}) axes. The local coordinate system of this joint is defined using the source axis.

You specify the initial source axis (\mathbf{e}_{10}) and initial destination axis (\mathbf{e}_{d10}) in terms of components along the axes of the selected coordinate system. The initial source axis must point toward the center of joint, whereas the initial destination axis must point away from the center of joint.

The process of creating a rotated source axis, rotated destination axis, and a corresponding local coordinate system is the same as for the hinge joint. The auxiliary axis is taken as the initial destination axis unless it is parallel to the initial source axis.

BALL JOINT FORMULATION

For a ball joint, the destination attachment is free to rotate relative to the source attachment about all three axes. The added degrees of freedom are contained in a relative quaternion (a , b , c , and d), which represents the rotation in 3D. The rest of the formulation is similar to that of the hinge joint.

Joint Elasticity in Ball Joint

Since only translations are constrained in a ball joint, the elastic degrees of freedom are

$$\mathbf{u}_{e1} = u_1 \mathbf{e}_1 + u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3$$

Therefore, there are only elastic forces and no elastic moments or coupling matrices.

$$\mathbf{F}_{s,e1} = \mathbf{k}_u \mathbf{u}_{e1}$$

$$\mathbf{F}_{d,e1} = \mathbf{c}_u \dot{\mathbf{u}}_{e1}$$

The total elastic joint forces can be written using the spring and damping forces as

$$\mathbf{F}_{e1} = -\mathbf{F}_{s,e1} - \mathbf{F}_{d,e1}$$

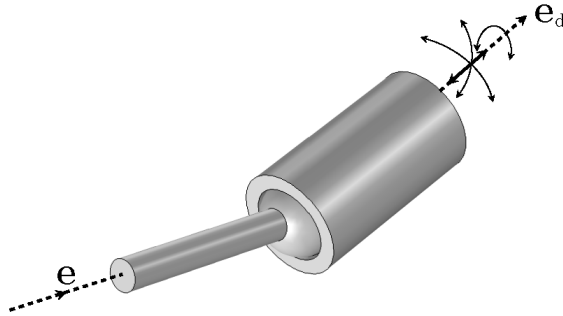
The contribution to the virtual work is

$$\delta W = \mathbf{F}_{e1} \cdot \delta \mathbf{u}_{e1}$$

Slot Joint Theory

The [Slot Joint](#) has three rotational and one translational degrees of freedom between the two attached components. The components are free to rotate relative to each other

about all three axes and also free to translate along the joint translation axis. The slot joint can be seen as a combination of the prismatic joint and the ball joint.



The following is an addition to the [Prismatic Joint Theory](#) and the [Ball Joint Theory](#), which are also applicable to the slot joint.

SLOT JOINT AXIS AND ITS LOCAL COORDINATE SYSTEM

For a slot joint, the translational motion is free along the joint translational axis, which can be either the source axis or the destination axis. The translational degrees of freedom is added along the joint translational axis.

You can specify the initial joint translational axis (\mathbf{e}_{tr0}) to be either

$$\mathbf{e}_{tr0} = \mathbf{e}_{10} \text{ or } \mathbf{e}_{tr0} = \mathbf{e}_{d10}$$

For the slot joint, the process of defining the source axis, destination axis, and local coordinate system is the same as for the ball joint.

The joint coordinate system is always attached to the source part even if the joint translational axis is defined by the destination axis.

SLOT JOINT FORMULATION

The slot joint formulation is similar to the ball joint except that an extra translational degree of freedom is needed. The joint translational axis is used for the relative displacement vector, and the rotational degrees of freedom are represented by a relative quaternion as in the ball joint.

Joint Elasticity in Slot Joint

The elastic degrees of freedom depends on whether the joint translational axis is defined to be the source axis or the destination axis. In both cases, there are two constrained translations, but it is only when the source axis is the joint translational axis that the elastic displacement can be expressed in the joint coordinate system direction

$$\mathbf{u}_{e1} = u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3$$

When the joint translational axis is the destination axis, then

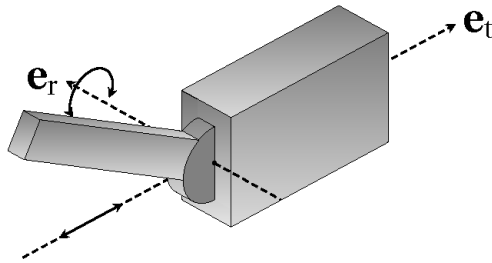
$$\mathbf{u}_{e1} = u_2 \mathbf{e}_{d2} + u_3 \mathbf{e}_{d3}$$

where \mathbf{e}_{d2} and \mathbf{e}_{d3} are second and third axis of the destination coordinate system, respectively. In this case, the stiffness and damping matrices are given along the destination axes when the joint coordinate system is selected.

Except for this difference, the formulation is the same as for the ball joint.

Reduced Slot Joint Theory

The [Reduced Slot Joint](#) has one rotational and one translational degree of freedom between the two attached components. Both components of the system are free to rotate about the joint rotational axis and free to translate along the joint translation axis. The reduced slot joint can be thought of as a combination of the prismatic joint and the hinge joint, with different axes for the allowed degrees of freedom. The translational axes and the rotational axis are orthogonal to each other.



The reduced slot joint in 2D is similar to the reduced slot joint in 3D. The only difference is the initial joint rotational axis ($\mathbf{e}_{\text{rot}0}$), which does not change in 2D and is always the out-of-plane axis (\mathbf{e}_z).



The following is an addition to the [Cylindrical Joint Theory](#), which is also applicable for the reduced slot joint.

REDUCED SLOT JOINT AXES AND ITS LOCAL COORDINATE SYSTEM

The translational motion for a reduced slot joint is free along the joint translational axis, which can either be attached on the source attachment or the destination attachment. The translational degree of freedom is added along the joint translational axis.

In addition to the translational axis, information about the joint rotational axis is also required. You specify its components in the selected coordinate system. In this joint, the initial joint axis (\mathbf{e}_{10}) and initial second axis (\mathbf{e}_{20}) are the same as the initial joint translational axis ($\mathbf{e}_{\text{tr}0}$) and initial joint rotational axis ($\mathbf{e}_{\text{rot}0}$), respectively. If these axes are not orthogonal on input, then the rotational axis is adjusted in the plane spanned by $\mathbf{e}_{\text{tr}0}$ and $\mathbf{e}_{\text{rot}0}$ so as to fulfill that criterion.

REDUCED SLOT JOINT FORMULATION

The joint formulation is similar to the cylindrical joint except that the translational degree of freedom is defined along the joint translation axis and the rotational degree of freedom is defined about the joint rotational axis.

Joint Elasticity in Reduced Slot Joint

The elastic degrees of freedom are written as

$$\mathbf{u}_{e1} = u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3$$

$$\theta_{e1} = \theta_1 \mathbf{e}_1 + \theta_3 \mathbf{e}_3$$

Fixed Joint Theory

The **Fixed Joint** rigidly connects the two components and does not allow any relative motion between them. It does not add any degrees of freedom.



The following is an addition to the **Prismatic Joint Theory**, some of which is also applicable for the fixed joint.

FIXED JOINT FORMULATION

Constraints are added stating that the both parts have the same value of displacements and rotations at the center of joint. In 3D six constraints are added:

$$\begin{aligned}\mathbf{u}_{c, \text{dst}} &= \mathbf{u}_{c, \text{src}} \\ \mathbf{b}_{\text{dst}} &= \mathbf{b}_{\text{src}}\end{aligned}$$

In 2D three constraints are added:

$$\begin{aligned}\mathbf{u}_{c, \text{dst}} &= \mathbf{u}_{c, \text{src}} \\ \phi_{\text{dst}} &= \phi_{\text{src}}\end{aligned}$$

The position of the center of joint only affects interpretation of the joint forces and moments.

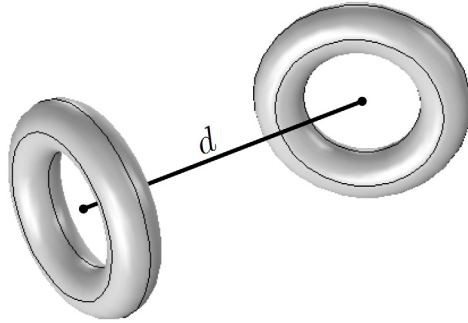
Joint Elasticity in Fixed Joint

The elastic degrees of freedom are written as

$$\begin{aligned}\mathbf{u}_{e1} &= u_1 \mathbf{e}_1 + u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3 \\ \theta_{e1} &= \theta_1 \mathbf{e}_1 + \theta_2 \mathbf{e}_2 + \theta_3 \mathbf{e}_3\end{aligned}$$

Distance Joint Theory

The **Distance Joint** maintains a specified distance between two points that are connected to a pair of attachments or rigid domains. As a default, the initial distance is maintained, so that the joint can be considered as a rigid link with ball joints at both ends. It is also possible to vary the distance so that the joint acts as a telescopic link.



The following is an addition to the [Prismatic Joint Theory](#), some of which is also applicable for the distance joint.

DISTANCE JOINT FORMULATION

The formulation of the distance joint differs completely from the formulation of the other joint types. Only a constraint is applied to maintain the distance between the specified points on both selected components. This distance constraint can be written as:

$$|(\mathbf{X}_{\text{src}} + \mathbf{u}_{\text{src}}) - (\mathbf{X}_{\text{dst}} + \mathbf{u}_{\text{dst}})| = |\mathbf{X}_{\text{src}} - \mathbf{X}_{\text{dst}}| + \Delta$$

Here \mathbf{X}_{src} and \mathbf{X}_{dst} are the undeformed locations of the two connection points, and \mathbf{u}_{src} and \mathbf{u}_{dst} are the corresponding displacements. It is possible to vary the distance between the points through the extension value Δ .

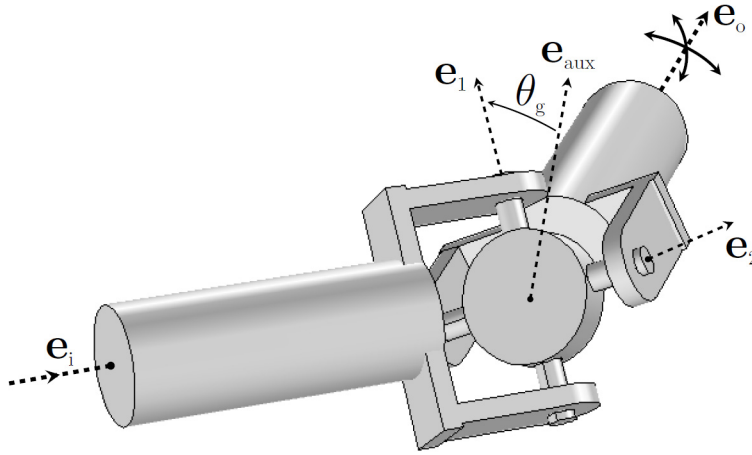
This constraint can be simplified and is used in the following form:

$$(\mathbf{u}_{\text{src}} - \mathbf{u}_{\text{dst}}) \cdot (\mathbf{u}_{\text{src}} - \mathbf{u}_{\text{dst}}) + 2(\mathbf{u}_{\text{src}} - \mathbf{u}_{\text{dst}}) \cdot (\mathbf{X}_{\text{src}} - \mathbf{X}_{\text{dst}}) = \Delta^2 + 2\Delta|\mathbf{X}_{\text{src}} - \mathbf{X}_{\text{dst}}|$$

Universal Joint Theory

An [Universal Joint](#), also known as Hooke's joint or Cardan joint, is commonly used to transmit rotational motion between shafts that have different orientations. Physically this joint is made up of a pair of hinge joints, oriented perpendicular to each other, connected by a cross shaft or gimbal. This joint is a kind of abstract joint where the

gimbal is not modeled and both the shafts are directly connected to each other. In general, this joint is used to transfer mechanical power from one shaft to another shaft. The added degrees of freedom in this joint are the relative rotations about the two legs of the gimbal that form the joint axis and second axis.



The following is an addition to the [Prismatic Joint Theory](#), some of which is also applicable for the universal joint.

JOINT AXIS AND LOCAL COORDINATE SYSTEM

In this joint, the initial orientation of input and output shafts are based on user inputs. The initial auxiliary axis is defined as perpendicular to the plane spanned by the shafts:

$$\mathbf{e}_{\text{aux}0} = \mathbf{e}_{i0} \times \mathbf{e}_{o0}$$

If both the shafts have the same initial orientation, the initial output shaft orientation in this expression is replaced with an arbitrary axis, which is not parallel to the initial input shaft orientation.

You can also define the auxiliary axis explicitly. In this case, a projection is made in the plane perpendicular to the initial input shaft orientation to ensure the orthogonality of initial auxiliary axis and initial input shaft orientation.

The initial axis connecting the gimbal and input shaft or initial joint axis can be defined as being the same as the auxiliary axis

$$\mathbf{e}_{10} = \mathbf{e}_{\text{aux}0}$$

Alternatively, the initial joint axis can be defined by rotating the initial auxiliary axis ($\mathbf{e}_{\text{aux}0}$) about the initial input shaft orientation (\mathbf{e}_{i0}) with a specified angle (θ_{g0}):

$$\mathbf{e}_{10} = \mathbf{e}_{\text{aux}0} \cos(\theta_{g0}) + (\mathbf{e}_{i0} \times \mathbf{e}_{\text{aux}0}) \sin(\theta_{g0}) + \mathbf{e}_{i0} (\mathbf{e}_{i0} \cdot \mathbf{e}_{\text{aux}0}) (1 - \cos(\theta_{g0}))$$

The initial second axis and initial third axis are formed by:

$$\mathbf{e}_{20} = \mathbf{e}_{10} \times \mathbf{e}_{o0}$$

$$\mathbf{e}_{30} = \mathbf{e}_{10} \times \mathbf{e}_{20}$$

The rotated local coordinate system computed as:

$$\mathbf{e}_1 = R_{\text{src}} \mathbf{e}_{10}$$

$$\mathbf{e}_{30} = R_{\text{dst}} \mathbf{e}_{20}$$

$$\mathbf{e}_3 = \mathbf{e}_1 \times \mathbf{e}_2$$

The current orientation of input and output shafts is computed by multiplying the initial directions with the corresponding rotation matrices:

$$\mathbf{e}_i = R_{\text{src}} \mathbf{e}_{i0}$$

$$\mathbf{e}_o = R_{\text{dst}} \mathbf{e}_{o0}$$

UNIVERSAL JOINT FORMULATION

The formulation is similar to that of a hinge joint, except the definition of the relative quaternion, which is written as

$$(a_r, \mathbf{b}_r) = (a_{r2}, \mathbf{b}_{r2}) \otimes (a_{r1}, \mathbf{b}_{r1})$$

where

$$a_{r1} = \cos(\theta_1/2) \quad \mathbf{b}_{r1} = \sin(\theta_1/2) \mathbf{e}_1$$

$$a_{r2} = \cos(\theta_2/2) \quad \mathbf{b}_{r2} = \sin(\theta_2/2) \mathbf{e}_2$$

Joint Elasticity in Universal Joint

The elastic degrees of freedom are written as

$$\begin{aligned}\mathbf{u}_{e1} &= u_1 \mathbf{e}_1 + u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3 \\ \theta_{e1} &= \theta_3 \mathbf{e}_3\end{aligned}$$

POSTPROCESSING VARIABLES

For the universal joint, some extra variables for use in postprocessing are defined.

Angular Velocity and Total Rotation of Input and Output Shafts

The angular velocities and total rotation for both input and output shafts are computed:

$$\begin{aligned}\omega_i &= \boldsymbol{\omega}_i \cdot \mathbf{e}_i \\ \omega_o &= \boldsymbol{\omega}_o \cdot \mathbf{e}_o \\ \theta_i &= \int \omega_i dt \\ \theta_o &= \int \omega_o dt\end{aligned}$$

In order to evaluate the integrals, two extra ODE degrees of freedom are created for each universal joint.

Torsional Moment in the Shafts

The torsional moments in the input and output shafts are computed using

$$\begin{aligned}M_i &= \mathbf{M}_i \cdot \mathbf{e}_i \\ M_o &= \mathbf{M}_o \cdot \mathbf{e}_o\end{aligned}$$

Angle Between Shafts

The angle between the shafts (bend angle) β is computed using

$$\beta = \text{acos}(\mathbf{e}_i \cdot \mathbf{e}_o)$$

Theory for Joint Constraints

Constraints are used to restrict the relative motion between the two components sharing a joint. You can specify the upper and lower bound of the relative motion. Applying constraints on the relative motion can be interpreted as placing an imaginary stopper that restricts the motion of the destination attachment with respect to the

source attachment in a prescribed direction. [Translational Constraints](#), [Rotational Constraints](#), [Inclination Angle Constraints](#), and [Axial Rotation Constraints](#) are discussed in this section.

TRANSLATIONAL CONSTRAINTS

Translational constraints are available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) features.

The maximum relative displacement (u_{\max}) and minimum relative displacement (u_{\min}) can be prescribed. The activation condition for the constraint is:

$$(u > u_{\max}) \vee (u < u_{\min})$$

A penalty method is used to enforce the constraints.

Penalty Method

The constraint on the relative displacement is enforced using a stiff spring between the components. The stiffness of the spring is defined by the penalty factor. When the constraint is activated in a dynamic analysis, the time steps are reduced in order to conserve the momentum in the system.

The default value of the penalty factor p_u is evaluated as:

$$p_u = \left(\frac{\frac{du}{dt}}{\delta u_{\max}} \right)^2 m_e \quad (3-1)$$

The numerator in [Equation 3-1](#) is an assumed relative velocity between the components when the constraint is applied, and the denominator is the maximum allowable penetration. This ratio of the relative velocity and the maximum allowable penetration decides the required stiffness of the spring (the penalty factor). The default value of this ratio is 10^3 s^{-1} . This could, for example, be a penetration of 1 mm with an impact speed of 1 m/s. You can change this ratio based on the dynamics of the system and the allowable constraint violation.



For very high penalty factors, the solution can be inaccurate unless the tolerances are tightened.

The factor m_e in the penalty factor expression is the effective mass at the joint, defined as

$$m_e = \frac{(m_{\text{src}} m_{\text{dst}})}{(m_{\text{src}} + m_{\text{dst}})}$$

The source and destination masses m_{src} and m_{dst} are defined as:

- The mass of the domains attached to the source or destination attachments in case of flexible components.
- The mass of the attached rigid domain in case of rigid components.

The following assumptions are made in the definition of the effective mass:

- Both parts are free to move. If only one of the parts is free, you should use the mass associated with that part as the effective mass.
- The computation of m_{src} and m_{dst} takes into account the domains that are directly adjacent to the source and destination attachment. These domains cannot be the only parts affecting the dynamics of an impact.
- The computation of m_{src} and m_{dst} also do not account for the mass added using Added Mass features or Mass and Moment of Inertia features.

The default value for the maximum time step allowed in order to capture the event accurately is evaluated as:

$$\Delta t_{\text{max}} = \frac{(T/2)}{(30)} \quad T = 2\pi \sqrt{\frac{m_e}{p_u}}$$

where T is the time period for free vibration of a spring-mass system having a stiffness equal to the penalty factor and mass equal to the effective mass. As can be seen from the expressions above, the default maximum time step is approximately 0.1 ms.

The constraint is violated for half of the period ($T/2$). To capture that phenomenon accurately, approximately 30 time steps are used. There is a tradeoff between accuracy and computation time. A larger number of steps reduces the maximum allowable time step and increases the accuracy in conserving the momentum. However, this also increases the computation time.

To prevent the solver from taking time steps larger than the maximum allowable time step, the penalty factor actually used is:

$$p_{\text{mu}} = p_u + ((u > u_{\text{max}}) || (u < u_{\text{min}})) \cdot \text{if}((\Delta t > \Delta t_{\text{max}}), \infty, 0)$$

This means that modified penalty factor p_{mu} is equal to the given penalty factor as long as the time step taken by the solver is smaller than the maximum allowable time step.

If the constraint is violated, and the time step taken by the solver is larger than the maximum allowable time step, the solver is forced to decrease the time step.



The penalty factor is not modified for quasi-static and stationary-parametric types of analyses because the conservation of momentum is not relevant in these cases.

The force required to constrain the relative motion is

$$F_u = \begin{cases} -p_{mu} \cdot (u - u_{\max}) & u > u_{\max} \\ -p_{mu} \cdot (u - u_{\min}) & u < u_{\min} \\ 0 & \text{else} \end{cases}$$

To enforce a constraint condition, a virtual work contribution is added when constraints are active:

$$\delta W = F_u \cdot \delta u$$

In the case of a [Planar Joint](#), the relative displacement is constrained along a user-defined axis (the *constraint direction*) oriented in a plane perpendicular to the joint axis. The relative displacement in the constraint direction, u_c , is:

$$u_c = (u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3) \cdot (R_{\text{src}} \mathbf{e}_c)$$

where \mathbf{e}_c is the normalized constraint axis given by the user.

ROTATIONAL CONSTRAINTS

Rotational constraints are available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#) features.

The theory for rotational constraints is similar to what is described above for [Translational Constraints](#).

Here, the penalty factor should be considered as a torsional spring. The effective mass in the default penalty factor is replaced by an effective moment of inertia, having an analogous definition.

INCLINATION ANGLE CONSTRAINTS

Inclination angle constraints are available for the [Ball Joint](#) and [Slot Joint](#) features.

Here, the relative rotation to which the constraint is applied is the inclination of the destination axis with respect to the reference axis (also called an inclination angle or polar angle). The reference axis can be chosen as the initial source axis, initial destination axis, or as a user-defined arbitrary axis. The reference axis is always following the source attachment.

The constraint is applied only on the upper limit of the inclination angle. The upper limit of the inclination angle should be less than 180° . The constraint on the inclination angle is independent of the azimuthal angle and cannot be a function of azimuthal angle.

You give the maximum relative inclination angle (θ_{\max}) as a limit on the inclination of the destination attachment with respect to the reference direction. The actual constraint is applied to the cosine of the inclination angle:

$$\cos\theta = \mathbf{e}_R \cdot \mathbf{e}_{d1}$$

where \mathbf{e}_R is the reference axis and \mathbf{e}_{d1} is the destination axis.

The activation condition for an inclination angle constraint is written as:

$$\cos\theta < \cos\theta_{\max}$$

The default value for the penalty factor and maximum allowable time step is computed as for [Rotational Constraints](#). The moments of inertia for the source and destination components are computed about an axis that is passing through the joint center and is perpendicular to both the source and destination axes.

In order to prevent the solver from taking time steps larger than the maximum allowable time step, a modified penalty factor is defined as:

$$p_{m\theta} = p_\theta + (\cos\theta < \cos\theta_{\max}) \cdot \text{if}((\Delta t > \Delta t_{\max}), \infty, 0)$$

The moment required to enforce the inclination angle constraint is made proportional to the difference in the cosines of the angles, rather than to the difference in the angles themselves. This avoids operating with inverse trigonometric functions.

$$M_\theta = \begin{cases} p_{m\theta} \cdot (\cos\theta - \cos\theta_{\max}) \cdot \sin\theta_{\max} & \cos\theta < \cos\theta_{\max} \\ 0 & \text{else} \end{cases}$$

To enforce the constraint condition, a virtual work contribution is added for the cosine of the inclination angle:

$$\delta W = M_\theta \delta\theta = -\frac{M_\theta}{\sin\theta} \cdot \delta(\cos\theta) \approx -\frac{M_\theta}{\sin\theta_{\max}} \cdot \delta(\cos\theta)$$

AXIAL ROTATION CONSTRAINTS

Axial rotation constraints are available for the [Ball Joint](#) and [Slot Joint](#) features.

The relative rotation, on which the constraint is applied, is the relative rotation of the destination attachment about its own axis.

The constraint can be applied as both the upper and lower limits on the relative rotation of the destination attachment about its own axis. This constraint is uniquely defined as long as the maximum relative rotation is less than 180 degrees.

You prescribe the maximum and minimum axial relative rotation of destination attachment Ψ_{\max} and Ψ_{\min} . The cosine of the relative axial rotation of the destination attachment is written as:

$$\cos\psi = 1 - 2 \cdot (\mathbf{e}_r \cdot \mathbf{e}_{d1})^2 \cdot (1 - a_r^2)$$

$$\mathbf{e}_r = \frac{1}{\sqrt{(b_r^2 + c_r^2 + d_r^2)}} \cdot \begin{bmatrix} b_r \\ c_r \\ d_r \end{bmatrix}$$

Here (a_r, b_r, c_r, d_r) represents the relative quaternion, and \mathbf{e}_r is the normalized relative rotation axis.

The activation conditions for the maximum and minimum relative axial rotation constraints are written as:

$$(\cos\psi < \cos\Psi_{\max}) \cdot (\mathbf{e}_r \cdot \mathbf{e}_{d1} > 0)$$

$$(\cos\psi < \cos\Psi_{\min}) \cdot (\mathbf{e}_r \cdot \mathbf{e}_{d1} < 0)$$

The default value computation for the penalty factor and maximum allowable time step is the same as for [Rotational Constraints](#). The moment of inertia for the source and destination components are computed about an axis that is passing through the center of joint and is parallel to the destination axis.

The modified penalty factor, constraint moment, and virtual work contribution are similar to the [Inclination Angle Constraints](#). The only change is in the constraint activation conditions.

The **Locking** feature is used to lock the relative motion between the two components connected to a joint. You specify limits on the relative motion in the form of an upper and lower bound. This can be interpreted as an imaginary object (like a snap hook) that locks one of the free relative degrees of freedom of the destination attachment when it reaches the limiting value.

The Locking feature differs from **Constraints** in a sense that the value of the particular joint degrees of freedom becomes constant for the rest of the analysis once it attains its prescribed limiting value.

The **Translational Locking**, **Rotational Locking**, **Inclination Angle Locking**, and **Axial Rotation Locking** types are discussed in this section.

TRANSLATIONAL LOCKING

Translational locking is available for the **Prismatic Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Slot Joint**, and **Reduced Slot Joint** features.

In this locking type, a lock is applied to the relative displacement (u). Once the lock is active, its relative displacement becomes constant at the prescribed limiting value and the relative velocity goes to zero. If lock conditions are active for all the joint degrees of freedom, then the source and destination attachment moves as a single object.

You prescribe the maximum (u_{\max}) and minimum (u_{\min}) limits of relative displacement. The activation condition for locking is:

$$(u > u_{\max}) \vee (u < u_{\min})$$

Locking is implemented using a penalty method where the relative velocity is forced to become zero for the rest of the time. There is a loss of energy during locking, which can be interpreted as the energy given to the imaginary object responsible for locking the degrees of freedom. Due to the locking phenomenon, stress waves are generated in the flexible components. You can add damping to the problem to suppress unwanted stress waves.

The locking indicators (i_{\max} and i_{\min}) are ODE variables used to ensure that the lock is active for the rest of the simulation. The values of the indicators are switched from zero to a nonzero value once the lock is active, through the following equations:

$$\frac{d}{dt}(i_{\text{umax}}) = (u > u_{\text{max}})$$

$$\frac{d}{dt}(i_{\text{umin}}) = (u < u_{\text{min}})$$

The force required to lock the relative motion is

$$F_{\mathbf{u}} = \begin{cases} -p_{\text{u1}} \cdot (u - u_{\text{max}}) - p_{\text{u2}} \cdot \frac{du}{dt} & i_{\text{umax}} > 0 \\ -p_{\text{u1}} \cdot (u - u_{\text{min}}) - p_{\text{u2}} \cdot \frac{du}{dt} & i_{\text{umin}} > 0 \\ 0 & \text{else} \end{cases}$$

where p_{u1} and p_{u2} are the penalty factors. p_{u1} can be interpreted as a spring constant and its default value is a function of an equivalent Young's modulus and geometric dimensions. p_{u2} is a damping coefficient and its default value is 10 ms times the value of p_{u1} . these penalty factors are inputs and can you can modify them as required.

The value of the penalty factor (p_{u2}) controls the rate of decay of the relative velocity. A large value gives a high rate of decay but can cause convergence problems.

To enforce the locking condition, a contribution is added to the virtual work:

$$\delta W = F_{\mathbf{u}} \cdot \delta u$$

In the case of a [Planar Joint](#), the relative displacement is locked along a user-defined axis (the *locking direction*) oriented in a plane perpendicular to the joint axis. The relative displacement in the locking direction, u_1 , is:

$$u_1 = (u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3) \cdot (\mathbf{R}_{\text{src}} \mathbf{e}_1)$$

where \mathbf{e}_1 is the normalized locking axis given by the user.

ROTATIONAL LOCKING

Rotational locking is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#) features.

The theory for rotational constraints is similar to what is described above for [Translational Locking](#).

Here, the penalty factor should be considered as a torsional spring.

INCLINATION ANGLE LOCKING

Inclination angle locking is available for the [Ball Joint](#) and [Slot Joint](#) features.

Here, the lock is applied on the inclination of the destination axis relative to the reference axis (the inclination angle or polar angle). The ODE variable used as locking indicator is computed from

$$\frac{d}{dt}(i_{\theta\max}) = (\cos\theta < \cos\theta_{\max})$$

The moment required to apply an inclination angle lock is written as

$$M_{\theta} = \begin{cases} \left(p_{\theta 1} \cdot (\cos\theta - \cos\theta_{\max}) + p_{\theta 2} \cdot \frac{d}{dt}\cos\theta \right) \cdot \sin\theta_{\max} & i_{\theta\max} > 0 \\ 0 & \text{else} \end{cases}$$

To apply the lock, virtual work contributions are added for the cosine of the inclination angle:

$$\delta W = M_{\theta} \delta\theta = -\frac{M_{\theta}}{\sin\theta} \cdot \delta(\cos\theta) \approx -\frac{M_{\theta}}{\sin\theta_{\max}} \cdot \delta(\cos\theta)$$

AXIAL ROTATION LOCKING

Axial rotation locking is available for the [Ball Joint](#) and [Slot Joint](#) features.

Here, the lock is applied to the relative rotation of the destination attachment about its own axis. There are two ODE variables used as locking indicator. They are updated using

$$\begin{aligned} \frac{d}{dt}(i_{\psi\max}) &= (\cos\psi < \cos\psi_{\max}) \cdot ((\mathbf{e}_r \cdot \mathbf{e}_{d1}) > 0) \\ \frac{d}{dt}(i_{\psi\min}) &= (\cos\psi < \cos\psi_{\min}) \cdot ((\mathbf{e}_r \cdot \mathbf{e}_{d1}) < 0) \end{aligned}$$

The moment required to enforce the lock on the axial rotation is:

$$M_\psi = \begin{cases} \left(p_{\psi 1} \cdot (\cos \psi - \cos \psi_{\max}) + p_{\psi 2} \cdot \frac{d}{dt} \cos \psi \right) \cdot \sin \psi_{\max} & i_{\psi \max} > 0 \\ \left(p_{\psi 1} \cdot (\cos \psi - \cos \psi_{\min}) + p_{\psi 2} \cdot \frac{d}{dt} \cos \psi \right) \cdot \sin \psi_{\min} & i_{\psi \min} > 0 \\ 0 & \text{else} \end{cases}$$

To apply the lock, a virtual work contribution is added for the cosine of the axial rotation of the destination boundaries:

$$\delta W = M_\psi \delta \psi = -\frac{M_\psi}{\sin \psi} \cdot \delta(\cos \psi) \approx \begin{cases} -\frac{M_\psi}{\sin \psi_{\max}} \cdot \delta(\cos \psi) & i_{\psi \max} > 0 \\ -\frac{M_\psi}{\sin \psi_{\min}} \cdot \delta(\cos \psi) & i_{\psi \min} > 0 \end{cases}$$

Theory for Springs and Dampers on Joints

You can use the [Spring and Damper](#) feature to apply a spring or a dashpot to act on the relative motion in a joint. A dashpot can be used to account for various kinds of losses at the joint. The spring can be given a predeformation so that the spring equilibrium position is not the same as the initial state.

The [Translational Spring and Damper](#), [Rotational Spring and Damper](#), and [Inclination Angle and Axial Rotation Spring and Damper](#) versions of this feature are discussed in this section.

TRANSLATIONAL SPRING AND DAMPER

The translational type of spring and damper is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) features.

The spring force, which is proportional to the relative displacement is

$$F_s = -k_u(u - u_0)$$

where k_u is the spring constant and u_0 is the predeformation.

The damping force, which is proportional to the relative velocity, is:

$$F_d = -c_u \dot{u}$$

where c_u is the damping coefficient.

The virtual work contribution is

$$\delta W = (F_s + F_d) \delta u$$

In the case of a [Planar Joint](#), springs and dashpots are attached along user-defined axes oriented in a plane perpendicular to the joint axis. The spring axis and dashpot axis can differ. The forces are proportional to the relative displacement along the spring axis (u_s) and relative velocity along the dashpot axis (u_{dp}), respectively:

$$\begin{aligned} u_s &= (u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3) \cdot (R_{src} \mathbf{e}_s) \\ v_{dp} &= (v_2 \mathbf{e}_2 + v_3 \mathbf{e}_3) \cdot (R_{src} \mathbf{e}_{dp}) \end{aligned}$$

where \mathbf{e}_s is the normalized spring axis and \mathbf{e}_{dp} is the normalized damper axis.

ROTATIONAL SPRING AND DAMPER

The rotational type of spring and damper is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#) features

The theory for rotational spring and damper is similar to what is described above for [Translational Spring and Damper](#). All translational quantities are replaced by corresponding angular quantities.

INCLINATION ANGLE AND AXIAL ROTATION SPRING AND DAMPER

The inclination angle and axial rotation types of spring and damper are available for the [Ball Joint](#) and [Slot Joint](#) features.

The spring and damper here can affect two types of relative motion—the inclination angle and the axial rotation of the destination attachment. The inclination angle and axial rotation are computed as:

$$\begin{aligned} \theta &= \text{acos}(\min(\max(\{\cos\theta\}, -1), 1)) \\ \psi &= \text{acos}(\min(\max(\{\cos\psi\}, -1), 1)) \cdot ((\mathbf{e}_r \cdot \mathbf{e}_{d1} > 0) - (\mathbf{e}_r \cdot \mathbf{e}_{d1} < 0)) \end{aligned}$$



The definitions of inclination angle and axial rotation, as well as the expressions of cosines, are discussed in the [Inclination Angle Constraints](#) and [Axial Rotation Constraints](#) sections, respectively.

As these rotations are computed using the inverse cosine function, they are unique as long as the magnitudes of the actual rotations (θ and Ψ) are less than 180° .

The spring and damping forces are for both cases similar to those used for a standard [Rotational Spring and Damper](#).

Theory for Prescribed Motion on Joints

With [Prescribed Motion](#) you can control the relative motion in a joint, either by prescribing the displacement or the velocity. This is useful when the degrees of freedom in the joint are not free but are a known function of time. This is common in the field of robotics, for example.

The [Translational Prescribed Motion](#) and [Rotational Prescribed Motion](#) versions of this feature are discussed in this section.

TRANSLATIONAL PRESCRIBED MOTION

Translational prescribed motion is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) features.

A constraint is added to prescribe the relative displacement in the joint. If no reaction forces are computed, then the constraint is enforced as a pointwise constraint:

$$u = u_p$$

where u_p is the prescribed relative displacement. If reaction force computation has been requested by selecting **Evaluate reaction forces**, then a weak constraint is used instead. An extra degree of freedom, R_F , is added for the reaction force. The contribution to the virtual work is

$$(u - u_p)\delta R_F + R_F\delta(u - u_p)$$

If a unidirectional constraint is used (by selecting **Apply reaction only on joint variables**), then the form of the virtual work is changed to

$$(u - u_p)\delta R_F + R_F\delta u$$

When the relative velocity is prescribed, an extra equation is added to find the relative displacement by integration of the relative velocity:

$$\frac{d}{dt}(u_p) = v_p$$

where v_p is the prescribed relative velocity and u_p is the velocity integration variable which then is used in the constraint. In case of a stationary study, the relative displacement in the joint is constrained to zero.

You can also set an activation condition on the prescribed motion constraints. Once the specified activation condition is fulfilled and deactivation indicator expression (i_{up}), a Boolean variable, is set to true, the constraints are removed from the system. This makes the joint degrees of freedom free that were prescribed before.

In the case of a [Planar Joint](#), the relative displacements or velocities are prescribed along two axes oriented in a plane perpendicular to the joint axis. The first axis is specified by the user, while the second axis is in the same plane, and orthogonal to the first axis. The relative displacement along prescribed motion axis and along its orthogonal direction can be written as:

$$\begin{aligned} u_{ep} &= (u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3) \cdot (R_{src} \mathbf{e}_p) \\ u_{ep2} &= (u_2 \mathbf{e}_2 + u_3 \mathbf{e}_3) \cdot (\mathbf{e}_1 \times R_{src} \mathbf{e}_p) \end{aligned}$$

where \mathbf{e}_p is the normalized prescribed motion axis.

ROTATIONAL PRESCRIBED MOTION

Rotational prescribed motion is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Reduced Slot Joint](#), [Ball Joint](#), and [Slot Joint](#) features.

The theory for rotational prescribed motion is similar to that of [Translational Prescribed Motion](#). The difference is that the rotation or angular velocity is prescribed instead of displacement or translational velocity.

For the [Ball Joint](#) and [Slot Joint](#) features, relative motion is possible in all three directions. You prescribe the motion using a rotation axis and the magnitude of rotation or angular velocity about this axis.

The following constraint is added to [Ball Joint](#) and [Slot Joint](#) features to prescribe the relative rotation in a joint:

$$\mathbf{b}_r = \sin\left(\frac{\phi_p}{2}\right) \cdot \mathbf{e}_p$$

where ϕ_p is the magnitude of the prescribed relative rotation, and \mathbf{e}_p is the axis of relative rotation.

If a relative angular velocity is prescribed, an extra equation is added to find the relative rotation by integration of the relative angular velocity:

$$\frac{d}{dt}(\phi_p) = \Omega_p$$

where Ω_p is the prescribed relative angular velocity, and ϕ_p is the angular velocity integration variable.



Evaluating reaction force for the prescribed motion by selecting **Evaluate reaction forces** in the **Reaction force settings** section of the settings for **Prescribed Motion** has several benefits over evaluating it using joint forces and moments computed using weak constraints:

- It includes the contributions of springs, dampers, and friction applied on the joint.
- It works even if there are redundant constraints in the system, especially in a rigid body system, since joint constraints cannot be applied in the weak form in such a system.
- It is computationally more efficient as it adds fewer degrees of freedom.

Theory for Applied Forces and Moments on Joints

The **Applied Force and Moment** feature is used to apply forces and moments on all types of joints. It can be applied at the source or destination joint attachment, or directly to the joint degrees of freedom.

When the load is applied directly to the joint degrees of freedom, the virtual work contributions added depend on the joint type:

- With scalar force F , for joints with one translational degree of freedom u (Prismatic, Cylindrical, Screw, Slot, Reduced Slot):

$$F\delta u$$

- With vector force \mathbf{F} , used in the Planar joint which has two translational degrees of freedom expanded into a global relative displacement vector \mathbf{u}_r :

$$\mathbf{F} \cdot \delta \mathbf{u}_r$$

This means that the component of \mathbf{F} which is orthogonal to the plane of the joint translation is removed.

- With scalar moment M , for joints with one rotational degree of freedom θ (Hinge, Cylindrical, Screw, Planar, Reduced Slot):

$$M\delta\theta$$

- With vector moment \mathbf{M} , for joints with three rotational degrees of freedom represented by the quaternion $\mathbf{a}_r, \mathbf{b}_r$ (Ball, Slot):

$$2(\{0, \mathbf{M}\} \otimes \{\mathbf{a}_r, \mathbf{b}_r\} \otimes \delta\{\mathbf{a}_r, \mathbf{b}_r\})$$



The theory for applying forces and moments on attachments is the same as described in the [Rigid Connector Theory](#) section in the *Structural Mechanics Module User's Guide*.



The links to sections described in external guides, such as the *Structural Mechanics Module User's Guide*, do not work in the PDF, only from the online help.

Theory for Friction in Joints

The [Friction](#) feature is used to add frictional loss to a joint. It is available for [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Ball Joint](#).

The friction force is modeled using a continuous friction law, which is capable of modeling sliding-sticking phenomena. A strict application of Coulomb's law involves discrete transition from sticking to sliding and vice versa, as dictated by a vanishing relative velocity. These discrete transitions cause numerical difficulties and to avoid them, the friction force is approximated with a continuous friction law:

$$F_f = -\mu N \frac{v}{|v|} \left(1 - e^{\frac{-|v|}{v_0}} \right)$$

where F_f is the friction force, μ is the frictional coefficient, N is the normal force, v is the slip velocity, and v_0 is the characteristic slip velocity. The term

$$\left(1 - e^{\frac{-|v|}{v_0}} \right)$$

is called the regularization factor.

The regularization factor smooths the friction force discontinuity. The characteristic slip velocity should be chosen to be small in comparison to the characteristic relative velocities encountered during the simulation. The continuous friction law describes both sliding and sticking behavior, that is, it completely replaces Coulomb's law. Sticking is replaced by creeping between the contacting bodies with a small relative velocity.

The exact interpretation of the friction model for each joint type is described in the following. In general, an expression for the friction force such as

$$F_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{u}}{|\dot{u}|} \left(1 - e^{-\frac{|\dot{u}|}{v_0}} \right)$$

is used. Here $F_{f,r}$ is the additional sliding resistance, $F_{f,\max}$ is the maximum friction force, and \dot{u} is the relative velocity in the joint. Since the joints are abstractions, there is no unique way to define the normal force N , to which the friction force is proportional. For each joint there are several available methods for computing N and you must select the most representative one.



Friction can only be applied to the joints in a time-dependent analysis.

PRISMATIC JOINT

Friction Force

The total friction force is defined as

$$F_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{u}}{|\dot{u}|} \left(1 - e^{-\frac{|\dot{u}|}{v_0}} \right)$$

The total normal force can be computed from joint forces, interference fit, or the combination of these two.

Normal Force Case 1: Contribution from Joint Forces

Case-1.1: Normal force computed from attachment reaction forces:

$$N = \sum |\mathbf{F}_a \cdot \mathbf{n}|$$

where \mathbf{F}_a are the reaction forces on the attachment boundaries, and \mathbf{n} is the surface normal in the deformed state.

Case-1.2: Normal force computed from joint forces and moments:

$$N = |\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}| + |\alpha_{11}M_{j11}| + |\alpha_{12}M_{j12}| + |\alpha_{13}M_{j13}| \quad (3D)$$

$$N = |\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}| + |\alpha M_{j13}| \quad (2D)$$

where \mathbf{F}_j are the joint forces, \mathbf{e} is the joint axis, M_{jli} are the components of the joint moment in joint local coordinate system, and α_i are contribution factors given by the user.

Case-1.3: Normal force computed through joint force in a specified direction.

$$N = |(\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}) \cdot (\mathbf{R}_{src} \mathbf{e}_{n0})|$$

where \mathbf{e}_{n0} is a direction given by the user. \mathbf{R}_{src} represents the rotation matrix for the rotation of the source attachment.



This method is available only in the prismatic joint.

Normal Force Case 2: Contribution from Interference Fit

Case-2.1: Normal force defined by pressure and contact area,

$$N = pA$$

where p is the contact pressure, and A is the contact area.

Case-2.2: Normal force defined by total force

$$N = F_n$$

where F_n is the total normal force.

Virtual Work and Power Loss

The virtual work contribution from the total friction force is obtained through a multiplication by the variation of the relative displacement between the two parts:

$$\delta W = F_f \cdot \delta u$$

The energy dissipation rate caused by friction can be written as

$$Q_f = -\mathbf{F}_f \cdot \dot{\mathbf{u}}$$

HINGE JOINT

Most of the theory is the same as for prismatic joints. Any differences are described here.

Friction Moment

The total friction moment is defined as:

$$M_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{\theta}}{|\dot{\theta}|} \left(1 - e^{-\frac{|\dot{\theta}|}{(v_0/r)}} \right) r$$

where $\dot{\theta}$ is the relative angular velocity in the joint, and r is the hinge radius, supplied by the user.

Normal Force

Case-1.2: Normal force computed through joint forces and moments:

$$N = |\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}| + |\alpha_{12}M_{j12}| + |\alpha_{13}M_{j13}| \quad (3D)$$

$$N = |\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}| \quad (2D)$$

Virtual Work and Power Loss

The virtual work contribution from the total friction moment is obtained through a multiplication by variation of the relative rotation between the two parts

$$\delta W = M_f \cdot \delta \theta$$

The energy dissipation rate caused by the friction moment can be written as

$$Q_f = -M_f \cdot \dot{\theta}$$

CYLINDRICAL JOINT

Most of the theory is the same as for the hinge joint. Any differences are described here.

Friction Force and Moment

The total friction force and total friction moment are defined as:

$$F_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \sin(\lambda) \left(1 - e^{\frac{-|\dot{u}|}{v_0}} \right)$$

$$M_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \cos(\lambda) \left(1 - e^{\frac{-|\dot{\theta}|}{(v_0/r)}} \right) r$$

$$\tan(\lambda) = \frac{\dot{u}}{r\dot{\theta}}$$

where r is the entered cylinder radius.

Virtual Work and Power Loss

The virtual work contributions of the total friction force and the total friction moment are obtained through a multiplication by the variations of the relative displacement and relative rotation between the two parts respectively.

$$\delta W = F_f \cdot \delta u + M_f \cdot \delta \theta$$

The energy dissipation rate caused by friction can be written as

$$Q_f = -F_f \cdot \dot{u} - M_f \cdot \dot{\theta}$$

SCREW JOINT

Most of the theory is same as for cylindrical joints. Any differences are described here.

Friction Force and Moment

The total friction force and total friction moment are defined as:

$$F_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \sin(\lambda) \frac{\dot{u}}{|\dot{u}|} \left(1 - e^{\frac{-|\dot{u}|}{v_0}} \right)$$

$$M_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \cos(\lambda) \frac{\dot{u}}{|\dot{u}|} \left(1 - e^{\frac{-|\dot{\theta}|}{(v_0/r)}} \right) r$$

$$\tan(\lambda) = \frac{np}{2\pi r}$$

where n is the number of starts, p is the pitch, and r is the screw radius.

Normal Force

Case-1.1: The normal force computed from attachment reaction forces is:

$$N = \sum |\mathbf{F}_a \cdot \mathbf{n}| + \sum |\mathbf{F}_a \cdot \mathbf{e}| \cos(\lambda)$$

Case-1.2: The normal force computed through joint forces and moments is:

$$N = |\mathbf{F}_j - (\mathbf{F}_j \cdot \mathbf{e})\mathbf{e}| + |\mathbf{F}_j \cdot \mathbf{e}| \cos(\lambda) + |\alpha_{12}M_{j12}| + |\alpha_{13}M_{j13}|$$

PLANAR JOINT

Most of the theory is same as for prismatic joints. Any differences are described here.

There are two ways to model the friction force in a planar joint. In the first approach, the total friction force is applied at the center of joint. This is similar to other joint types, but in this approach, no friction moment is caused by the relative rotation. In the second approach, the distributed friction forces are applied on the selected boundaries, and hence there is an effective friction moment for relative rotation.

Friction Force: Total Force at Center of Joint

The total friction force is defined as:

$$\mathbf{F}_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{\mathbf{u}}_r}{|\dot{\mathbf{u}}_r|} \left(1 - e^{-\frac{|\dot{\mathbf{u}}_r|}{v_0}} \right)$$

where $\dot{\mathbf{u}}_r$ is the relative velocity vector in the joint.

Friction Force: Distributed Force on Selected Boundaries

The distributed friction force is defined as:

$$\mathbf{f}_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{\mathbf{u}}_{rt}}{|\dot{\mathbf{u}}_{rt}|A} \left(1 - e^{-\frac{|\dot{\mathbf{u}}_{rt}|}{v_0}} \right)$$

$$\mathbf{u}_{rt} = \mathbf{u}_r + (R_{dst} - R_{src})(\mathbf{X} - \mathbf{X}_c)$$

where \mathbf{X} is the material coordinates on the selected boundaries, \mathbf{X}_c is the center of joint, and A is the total contact area.

Normal Force

Case-1.2: The normal force computed through joint forces and moments is:

$$N = |\mathbf{F}_j \cdot \mathbf{e}| + |\alpha_{12}M_{j12}| + |\alpha_{13}M_{j13}|$$

Virtual Work and Power Loss: Total Force at Center of Joint

The virtual work contribution from the total friction force is obtained through a multiplication by the variation of the relative displacement vector between the two parts:

$$\delta W = \mathbf{F}_f \cdot \delta \mathbf{u}_r$$

The energy dissipation rate caused by friction can be written as

$$Q_f = -\mathbf{F}_f \cdot \dot{\mathbf{u}}_r$$

Virtual Work and Power Loss: Distributed Force on Selected Boundaries

The virtual work contribution of the distributed friction force is obtained through a multiplication by the variation of the relative displacement vector between the two parts on the selected boundaries.

$$\delta W = \int \mathbf{f}_f \cdot \delta \mathbf{u}_{rt} dA$$

The energy dissipation rate caused by friction can be written as:

$$Q_f = \int \mathbf{f}_f \cdot \dot{\mathbf{u}}_{rt} dA$$

BALL JOINT

Most of the theory is same as in for hinge joints. Any differences are described here.

Friction Moment

The total friction moment is defined as

$$\mathbf{M}_f = -\min(\mu N + F_{f,r}, F_{f,\max}) \frac{\dot{\theta}}{|\dot{\theta}|} \left(1 - e^{-\frac{|\dot{\theta}|}{(v_o/r)}} \right) \frac{\pi r}{4}$$

where $\dot{\theta}$ is the relative angular velocity vector at joint and r is the ball radius. The factor $\pi/4$ is the effective ball radius.

Normal Force

Case-1.2: The normal force computed through joint forces is:

$$N = |\mathbf{F}_j|$$

Virtual Work and Power Loss

The virtual work contribution from the total friction moment is obtained through a multiplication by the variation of the relative rotation between the two parts,

$$\delta W = 2 \{0, \mathbf{M}_f\}^T \otimes \{a_r, \mathbf{b}_r\}^T \cdot \{\delta a_r, \delta \mathbf{b}_r\}^T$$

where $\{a_r, \mathbf{b}_r\}$ is the relative quaternion in the joint.

The energy dissipation rate caused by friction can be written as

$$Q_f = -\mathbf{M}_f \cdot \dot{\theta}$$

Theory for Joint Forces and Moments Computation

The joint forces and moments are evaluated in the global spatial coordinate system as well as in a joint local coordinate system. The joint forces and moments are referred to the center of joint in the current position. For the joints that have translational degrees of freedom, the center of joint should be interpreted as fixed to the source side.

SUMMING REACTION FORCES OVER THE BOUNDARIES

This is the default method for computing the joint forces and moments when flexible domains are attached to the joint. The computation can be done on either the source or destination attachment. The convention is that the force and moment are the ones acting on the attachment from the joint. The sign changes if you switch between using the source or destination attachment for the evaluation.

The joint forces are computed by integrating (that is, a summation over all nodes) the reaction forces on the attached boundaries.

$$\mathbf{F} = \sum_i^{nodes} \mathbf{F}_i$$

The joint moments are computed first by integrating the cross product of reaction forces and position vector of each node relative to the centroid of the attached boundaries and then transferring it to the center of the joint.

$$\mathbf{M}_a = \sum_i^{\text{nodes}} (\mathbf{R}_a (\mathbf{x}_i - \mathbf{x}_a)) \times \mathbf{F}_i$$

$$\mathbf{M} = \mathbf{M}_a + ((\mathbf{x}_a + \mathbf{u}_a) - (\mathbf{x}_c + \mathbf{u}_{c,src})) \times \mathbf{F}$$

Here the subscript 'a' indicates the centroid of attachment, and 'c' indicates the center of joint. \mathbf{R}_a is the rotation matrix representing the rotation of the attachment, and \mathbf{u} represents the displacements.

USING WEAK CONSTRAINTS

When using weak constraints, the joint constraints are applied in a weak form, and the values of the Lagrange multipliers give the joint forces (\mathbf{F}_w) and moments (\mathbf{M}_w). This works for both flexible and rigid components. The use of Lagrange multipliers can, however, have an effect on the structure of the equation system, which limits the solvers that can be used.

The joint forces and moments are computed as

$$\mathbf{F} = \mathbf{F}_w$$

$$\mathbf{M} = \mathbf{M}_w + (\mathbf{u}_{c,dst} - \mathbf{u}_{c,src}) \times \mathbf{F}_w$$

The correction term for the moment appears in joints that have translational degrees of freedom because \mathbf{M}_w is computed on the destination attachment, which can have moved relative to the source attachment.

The sign convention in this case is that the forces and moments should be interpreted as acting on the destination attachment.

USING PENALTY FORMULATION

When using penalty method, the joint constraints are applied using a penalty formulation, and the values of the penalty forces (\mathbf{F}_p) and moments (\mathbf{M}_p) give the joint forces and moments. This works for both flexible and rigid components. Moreover, this also works for overconstrained rigid-body systems in which the weak constraints method may not work due to an insufficient number of equations for the Lagrange multipliers.

Since the value of penalty factor directly affects the joint constraints, it should be chosen judiciously. The penalty factor value should be large enough to ensure that the joint constraints are not violated beyond a permissible limit. At the same time the penalty factor should not be too high, since that will make the system of equations numerically ill-conditioned.

The penalty forces (\mathbf{F}_p) and moments (\mathbf{M}_p) are computed as

$$\begin{aligned}\mathbf{F}_p &= p_j \cdot \mathbf{c}_u \\ \mathbf{M}_p &= (0.1d)^2 p_j \cdot \mathbf{c}_\theta\end{aligned}$$

where \mathbf{c}_u and \mathbf{c}_θ are the translational and rotational violations of the joint constraints, respectively. The penalty factor p_j is the input having a default value, $0.1dE_{\text{equ}}$. In this expression, E_{equ} is an effective Young's modulus, and d is the geometric diagonal of the bounding box of the geometry.

The computation of joint forces and moments using penalty forces and moments is as described above for the weak constraints method:

$$\begin{aligned}\mathbf{F} &= \mathbf{F}_p \\ \mathbf{M} &= \mathbf{M}_p + (\mathbf{u}_{c, \text{dst}} - \mathbf{u}_{c, \text{src}}) \times \mathbf{F}_p\end{aligned}$$

Cam-Follower Theory

In this section:

- [Cam](#)
- [Follower](#)
- [Closest Point Determination](#)
- [Cam with Boundary Selection](#)
- [Cam with Edge Selection](#)
- [Cam-Follower Connection Force Computation](#)

Cam

A cam is defined through a set of boundaries or edges that is being followed by a point called “follower”. The cam selection can be defined on rigid as well as on elastic bodies. Thus, cam boundaries or edges can undergo any kind of rigid body motion as well as deformation. When defining a cam selection, it should also be possible to use boundaries or edges that are defined outside the physics interface.

Follower

A follower is defined through a point that follows the cam boundaries or edges with certain optional offset. The following options are supported to define a follower point:

- Centroid of selected entities (domains, boundaries, edges, or points)
- Geometric point (single point)

The follower point can be a part of rigid or elastic bodies. In case there is a requirement of multiple points to follow the cam boundaries or edges, multiple instances of this feature should be used.

Closest Point Determination

A general extrusion operator is defined in order to find a point on the cam boundaries or edges that is closest to the follower point. The general extrusion operator used has the following settings:

- Source selection: Cam boundaries or edges

- Destination map: Spatial position of the follower point
- Source frame: Spatial frame
- Mesh search method: Closest point

Cam with Boundary Selection

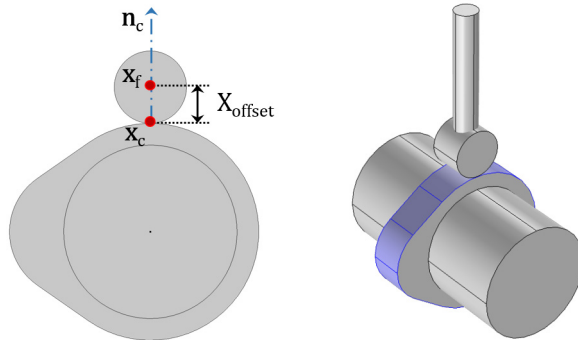


Figure 3-1: Example of a cam-follower with cam having boundary selection.

CONNECTION CONSTRAINT

The following constraint is added in order to impose the cam and follower connection:

$$\text{gap} = 0$$

In case the connection is conditionally active, the constraint expression is modified to the following:

$$\text{gap} \cdot (\text{slip} \leq \Delta) = 0$$

Here Δ is the connection tolerance. The gap and slip variables used in the connection constraint are defined in the following section.

GAP AND SLIP DEFINITION

The gap variable is a measure of distance between the follower point and its closest point on cam boundaries in the spatial normal direction.

The slip variable is a measure of distance between the follower point and its closest point on cam boundaries. Hence it is used to activate or deactivate connection constraint based on the specified connection tolerance.

The gap and slip between the follower point and cam boundaries can be defined as:

$$\text{gap} = (\mathbf{x}_f - \mathbf{x}_c) \cdot \mathbf{n}_c - X_{\text{offset}}$$

$$\text{slip} = \|\mathbf{x}_f - \mathbf{x}_c\| - X_{\text{offset}}$$

Here

- \mathbf{x}_f is the spatial position of the follower point.
- \mathbf{x}_c is the spatial position of a closet point on the cam boundaries.
- \mathbf{n}_c is the spatial normal of a closet point on the cam boundaries.
- X_{offset} is the initial offset/gap between the follower point and cam boundaries in the normal direction.

Cam with Edge Selection

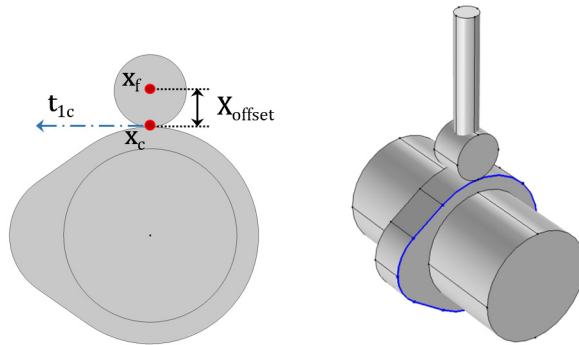


Figure 3-2: Example of a cam-follower with cam having edge selection.

CONNECTION CONSTRAINT

The following constraint is added in order to impose the cam and follower connection:

$$\mathbf{gap} = \mathbf{0}$$

In case the connection is conditionally active, the constraint expression is modified to the following:

$$\mathbf{gap} \cdot (\text{slip} \leq \Delta) = \mathbf{0}$$

Here Δ is the connection tolerance. The **gap** and **slip** used in the connection constraint are defined in the following section.

GAP AND SLIP DEFINITION

The **gap** variable is a measure of displacement vector between the follower point and its closest point on cam boundaries.

The **slip** variable is a measure of distance between the follower point and its closest point on cam boundaries in spatial tangent direction. Hence it is used to activate or deactivate connection constraint based on the specified connection tolerance.

The **gap** and **slip** between the follower point and cam edges can be defined as:

$$\mathbf{x}_{\text{offset}} = \mathbf{R}_c \cdot (\mathbf{X}_{\text{offset}} - (\mathbf{X}_{\text{offset}} \cdot \mathbf{t}_{10c})\mathbf{t}_{10c})$$

$$\text{slip} = (\mathbf{x}_f - \mathbf{x}_c) \cdot \mathbf{t}_{1c}$$

$$\mathbf{gap} = (\mathbf{x}_f - \mathbf{x}_c) - \mathbf{x}_{\text{offset}} - \text{slip} \cdot \mathbf{t}_{1c}$$

Here

- \mathbf{x}_f is the spatial position of the follower point.
- \mathbf{x}_c is the spatial position of a closet point on the cam edges.
- \mathbf{t}_{1c} is the spatial tangent of a closet point on the cam edges.
- $\mathbf{X}_{\text{offset}}$ is the initial offset or gap vector between the follower point and cam edges.
- \mathbf{t}_{10c} is the material tangent of a closet point on the cam edges.
- \mathbf{R}_c is the rotation matrix of the cam.

Cam-Follower Connection Force Computation

The cam-follower connection constraints can be imposed in several forms:

- Pointwise constraint
- Weak constraint
- Penalty method

The connection constraint enforces a continuous contact between the cam and follower; however, in order to find out whether the follower actually remains

continuously in contact with the cam, the connection force can be computed. A negative value of the connection force throughout the rotation of cam indicates the continuous contact between the cam and follower.

The pointwise constraint is the default form of the cam-follower connection constraints. In this form of constraint, the reaction forces representing the connection forces are not computed. In order to find connection forces, weak constraints or a penalty method can be used.

USING WEAK CONSTRAINTS

When using weak constraints, the connection constraints are applied in a weak form, and the values of the Lagrange multipliers give the connection forces (\mathbf{F}_w). The use of Lagrange multipliers can, however, have an effect on the structure of the equation system, which limits the solvers that can be used.

USING PENALTY FORMULATION

When using penalty method, the connection constraints are applied using penalty formulation, and the values of the penalty forces (\mathbf{F}_p) give the connection forces.

Since the value of penalty factor directly affects the connection constraints, it should be chosen judiciously. The penalty factor value should be large enough to ensure that the connection constraints are not violated beyond a permissible limit. At the same time the penalty factor should not be too high, because that will make the system of equations numerically ill-conditioned.

The penalty forces (\mathbf{F}_p) are computed as

$$\mathbf{F}_p = p_c \cdot \mathbf{gap}$$

where p_c is the penalty factor input having a default value, $0.1dE_{\text{equ}}$. In this expression, E_{equ} is an effective Young's modulus, and d is the geometric diagonal of the bounding box of the geometry.

Spring-Damper Theory

In this section:

- Connection Points and Spring Length
- Deactivation
- Spring and Damping Forces
- Spring and Damping Energies

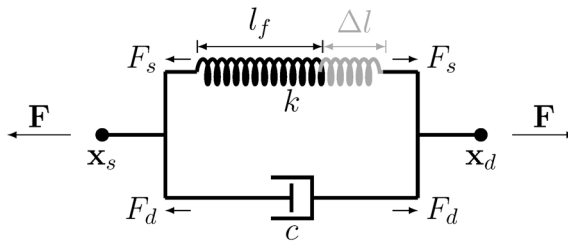


Figure 3-3: Conceptual sketch of a Spring-Damper.

Connection Points and Spring Length

The current position of the source and destination points, \mathbf{x}_s and \mathbf{x}_d , can be written as

$$\begin{aligned}\mathbf{x}_s &= \mathbf{X}_s + \mathbf{u}_s \\ \mathbf{x}_d &= \mathbf{X}_d + \mathbf{u}_d\end{aligned}$$

where \mathbf{X}_s and \mathbf{X}_d are the original positions of the two points, and \mathbf{u}_s and \mathbf{u}_d are their respective displacements. For the special case where the source point is selected as a fixed point,

$$\mathbf{x}_s = \mathbf{X}_s$$

The initial spring length, l_0 , is

$$l_0 = \|\mathbf{X}_d - \mathbf{X}_s\| = \sqrt{(\mathbf{X}_d - \mathbf{X}_s) \cdot (\mathbf{X}_d - \mathbf{X}_s)}$$

The current spring length, l , is

$$l = \|\mathbf{x}_d - \mathbf{x}_s\|$$

In the case of a geometrically linear analysis, the current spring length is linearized to

$$l = \frac{(\mathbf{x}_d - \mathbf{x}_s) \cdot (\mathbf{X}_d - \mathbf{X}_s)}{l_0}$$

In addition to the initial geometrical distance between the two points you can specify an initial spring extension Δl_0 , so that the free length of the spring is

$$l_f = l_0 - \Delta l_0$$

You can also specify the free length of the spring explicitly.

The spring extension Δl is computed as the difference between the current spring length and the free length,

$$\Delta l = l - l_f$$

Deactivation

The Spring-Damper can be deactivated under certain conditions. In terms of implementation, this means that many expressions are multiplied by an *activation indicator*, i_{ac} . The activation indicator has the value 1 when the component is active, and 0 when deactivated.

Spring and Damping Forces

The spring force is proportional to the spring constant k :

$$F_s = k \Delta l$$

If k depends on the extension, so that the spring is nonlinear, it should be interpreted as a secant stiffness, that is

$$F_s = k(\Delta l) \cdot \Delta l$$

You can also specify the spring force as function of extension explicitly, as

$$F_s = F_s(\Delta l)$$

To create the expression for the function, use the built-in variable for the spring extension. It has the form `<physicsTag>.<SpringNodeTag>.dl`, for example `mbd.spd1.dl`.

In a dynamic analysis, the viscous damping force is computed as

$$F_d = c \frac{d}{dt}(\Delta l)$$

where c is the viscous damping coefficient.

The magnitude of the total force is

$$F = F_s + F_d$$

The total forces in the global coordinate system, acting on the destination and source points are

$$\mathbf{F}_{dst} = -\mathbf{F}_{src} = -F \frac{(\mathbf{x}_d - \mathbf{x}_s)}{l}$$

In a geometrically linear case, the orientation of the force is kept fixed, so that

$$\mathbf{F}_{dst} = -\mathbf{F}_{src} = -F \frac{(\mathbf{X}_d - \mathbf{X}_s)}{l}$$

The contribution to the virtual work is

$$\mathbf{F}_{dst} \cdot (\delta \mathbf{u}_d - \delta \mathbf{u}_s)$$

Spring and Damping Energies

In stationary and time-dependent analysis, the elastic energy in the spring is computed as

$$W_s = i_{ac} \int_0^{\Delta l} F_s d(\Delta l)$$

In a time-dependent analysis, the energy dissipated in the damper, W_d , is computed using an extra degree of freedom. The following equation is added:

$$\frac{dW_d}{dt} = i_{ac} F_d \frac{d(\Delta l)}{dt}$$

In a frequency domain analysis, the elastic energy in the spring and the energy dissipated in the damper are computed as

$$W_s = \frac{1}{4} i_{ac} \operatorname{Re}(\operatorname{lindev}(F_s) \cdot \operatorname{conj}(\operatorname{lindev}(\Delta l)))$$

$$W_d = \frac{1}{4} i_{ac} \operatorname{Re}(\operatorname{lindev}(F_d) \cdot \operatorname{conj}(\operatorname{lindev}(\Delta l)))$$

These energy quantities represent the cycle average, and only the perturbation terms are included.

Gear Theory

In this section:

- Spur Gear Theory
- Helical Gear Theory
- Bevel Gear Theory
- Worm Gear Theory
- Spur Rack Theory
- Helical Rack Theory

Spur Gear Theory

The **Spur Gear** is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the **Gear Pair** node while connecting to other gears. Similar to the rigid domain, the spur gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

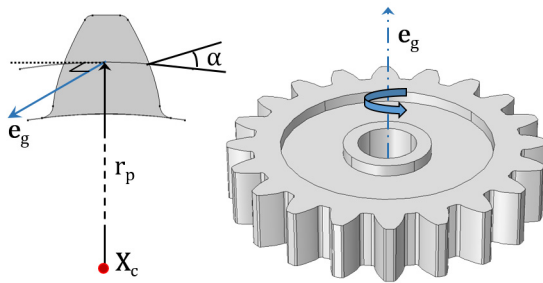


Figure 3-4: Sketch of an external spur gear., showing the tooth cross section and various parameters, such as center of rotation, gear axis, pitch radius, and pressure angle.

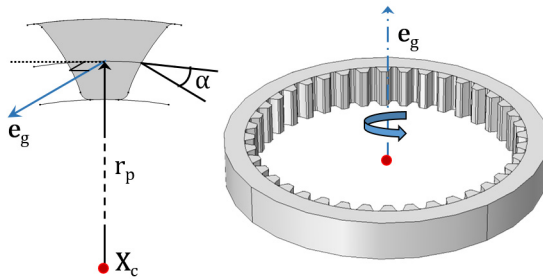


Figure 3-5: Sketch of an internal spur gear, showing the tooth cross section and various parameters such as center of rotation, gear axis, pitch radius, and pressure angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) section in the *Structural Mechanics Module User's Guide*.

CONNECTION TO OTHER GEARS

A spur gear can be connected to the following gear types:

- Another [Spur Gear](#) or a [Helical Gear](#) through the [Gear Pair](#) node.
- A [Worm Gear](#) through the [Worm and Wheel](#) node.
- A [Spur Rack](#) or a [Helical Rack](#) through the [Rack and Pinion](#) node.

A single gear can be connected to one or more gears.

MOUNTING METHODS

The motion of a spur gear can be constrained directly using its subnode. Alternatively, it can be mounted on a shaft. There are several ways through which a gear can be mounted on a shaft. Some of the commonly used methods are as follows:

- Directly mounted on a flexible shaft
- Mounted on a rigid or flexible shaft through a rigid or elastic fixed joint
- Mounted on a rigid or flexible shaft through a rigid or elastic hinge joint
- Mounted on a ground (a fixed shaft) through a rigid or elastic hinge joint

CENTER OF ROTATION

Gears are defined through rigid body degrees of freedom. These degrees of freedom are created at the center of rotation and the rotation is interpreted about this point. By

default the center of rotation is set to the center of mass, but there are other ways to define it explicitly. This is the point where you interpret the forces and moment that act on the gear due to meshing with other gears.

GEAR AXIS

The gear axis is the axis of rotation of the gear that passes through the center of rotation. This axis is used to create the gear's local coordinate system in the Gear Pair node. The gear rotation, a degree of freedom in the Gear Pair node, is also interpreted about this axis.

GEAR PROPERTIES

The following quantities are required to define a spur gear:

- Number of teeth (n)
- Pitch diameter (d_p)
- Pressure angle (α)

The transverse module of the gear is defined as:

$$m_t = \frac{d_p}{n}$$

The normal module of the gear is defined as:

$$m_n = m_t$$

Helical Gear Theory

The **Helical Gear** model is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the **Gear Pair** node while connecting to other gears. Similar to the rigid domain, the helical gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

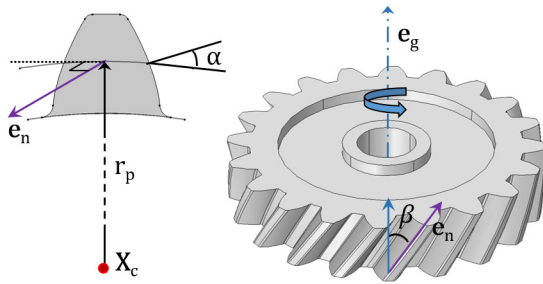


Figure 3-6: Sketch of an external helical gear, showing the tooth cross section and various parameters such as center of rotation, gear axis, pitch radius, pressure angle, and helix angle.

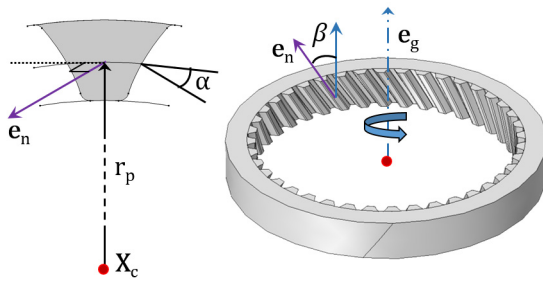


Figure 3-7: Sketch of an internal helical gear, showing the tooth cross section and various parameters such as center of rotation, gear axis, pitch radius, pressure angle, and helix angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) section in the *Structural Mechanics Module User's Guide*.



The following section is an addition to the [Spur Gear Theory](#) section, which is also applicable for the helical gear.

GEAR PROPERTIES

The additional parameter helix angle (β) is required to define a helical gear. The positive helix angle is interpreted as right handed and the negative helix angle is interpreted as left handed.

The normal module of the gear is defined as:

$$m_n = m_t \cos \beta$$

Bevel Gear Theory

The **Bevel Gear** model is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the **Gear Pair** node while connecting to other gears. Similar to the rigid domain, the bevel gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

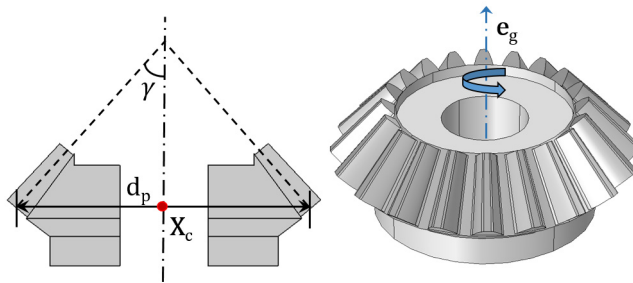


Figure 3-8: Sketch of a bevel gear, showing the gear cross section and various parameters such as center of rotation, gear axis, pitch diameter, and cone angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) in the *Structural Mechanics Module User's Guide*.



The following section is an addition to the [Spur Gear Theory](#) section, some of which is also applicable for the bevel gear.

CONNECTION TO OTHER GEARS

A bevel gear can be connected to another **Bevel Gear** through the **Gear Pair** node. A single gear can be connected to one or more gears.

GEAR PROPERTIES

The additional parameter cone angle (γ) is required to define a bevel gear. The positive cone angle results in a decrease in diameter along the gear axis.

Worm Gear Theory

The **Worm Gear** model is defined as a rigid body. A finite stiffness for the gear mesh (or gear tooth) can be specified in the **Worm and Wheel** node while connecting to a wheel. Similar to the rigid domain, the worm gear is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

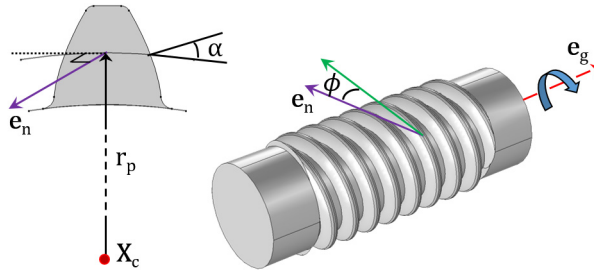


Figure 3-9: Sketch of a worm gear, showing the tooth cross-section and various parameters such as center of rotation, gear axis, pitch radius, pressure angle, and lead angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) section in the *Structural Mechanics Module User's Guide*.



The following section is an addition to the [Spur Gear Theory](#), section some of which is also applicable for the worm gear.

CONNECTION TO OTHER GEARS

A worm gear can be connected to a [Helical Gear](#), or [Spur Gear](#) through the [Worm and Wheel](#) node. A single gear can be connected to one or more gears.

GEAR PROPERTIES

The following quantities are required on define a worm gear.

- Number of starts (n)
- Pitch diameter (d_p)
- Pressure angle (α)

- Lead angle (ϕ)
- Handedness: Right or left

The helix angle for the right handed worm gear is defined as:

$$\beta = \frac{\pi}{2} - \phi$$

The helix angle for the left handed worm gear is defined as:

$$\beta = -\left(\frac{\pi}{2} - \phi\right)$$

The transverse module of the gear is defined as:

$$m_t = \frac{d_p}{n}$$

The normal module of the gear is defined as:

$$m_n = m_t \cos \beta$$

Spur Rack Theory

The **Spur Rack** or straight rack is defined as a rigid body. A finite stiffness for the rack mesh (or rack tooth) can be specified in the **Rack and Pinion** node while connecting to a pinion. Similar to the rigid domain, the spur rack is a material model, which is

mutually exclusive to all other material models. The only material property needed is the mass density.

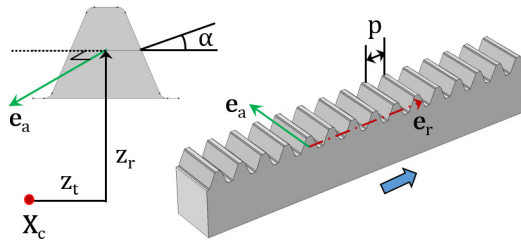


Figure 3-10: Sketch of a spur rack, showing the tooth cross-section and various parameters such as center of rotation, rack axis, pitch, radial offset, tangential offset, and pressure angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) section in the *Structural Mechanics Module User's Guide*.



The following section is an addition to the [Spur Gear Theory](#) section, some of which is also applicable for the spur rack.

CONNECTION TO OTHER GEARS

A spur rack can be connected to a [Spur Gear](#) or to a [Helical Gear](#) through the [Rack and Pinion](#) node. A single rack can be connected to one or more gears.

RACK AXIS

The rack axis is the axis of translation of the rack that passes through the center of rotation. This axis is used to create the rack local coordinate system in the rack and pinion node. The rack displacement, a degree of freedom in the rack and pinion node, is also interpreted about this axis.

RACK PROPERTIES

The following quantities are required to define a spur rack.

- Pitch (p)
- Pressure angle (α)

The normal pitch of the rack is defined as:

$$P_n = P$$

Helical Rack Theory

The **Helical Rack** is defined as a rigid body. A finite stiffness for the rack mesh (or rack tooth) can be specified in the **Rack and Pinion** node while connecting to a pinion. Similar to the rigid domain, the helical rack is a material model, which is mutually exclusive to all other material models. The only material property needed is the mass density.

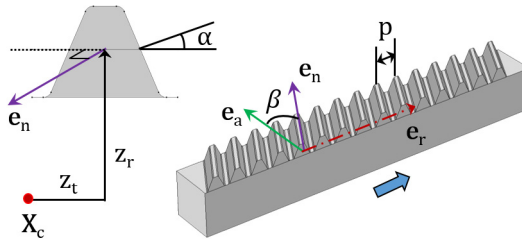


Figure 3-11: Sketch of a helical rack, showing the tooth cross-section and various parameters such as center of rotation, rack axis, pitch, radial offset, tangential offset, pressure angle, and helix angle.



For details about the theory of a rigid body, see the [Rigid Domain Model](#) section in the *Structural Mechanics Module User's Guide*.



The following section is an addition to the [Spur Rack Theory](#) section, which is also applicable for the helical rack.

RACK PROPERTIES

The additional parameter helix angle (β) is required to define a helical rack.

The normal pitch of the rack is defined as:

$$p_n = p \cos \beta$$

Gear Pair Theory

In this section:

- Theory for Gear Pairs
- Theory for Worm and Wheel
- Theory for Rack and Pinion
- Theory for Gear Elasticity
- Theory for Transmission Error
- Theory for Backlash
- Theory for Gear Friction

Theory for Gear Pairs

The **Gear Pair** node connects two spur gears, helical gears, or bevel gears in such a way that at the contact point, they have no relative motion along the line of action. The remaining displacements and rotations of both gears are independent of each other.

In case of a line contact model, one additional constraint is added to restrict the relative rotation about a line joining the two gear centers. When friction is included for a gear pair, frictional forces in the plane perpendicular to the line of action are added on both the gears.

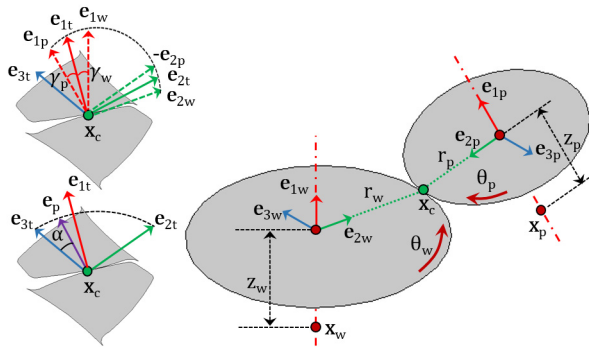


Figure 3-12: Sketch of a gear pair where the line of action is computed by the clockwise rotation of the tangent.

The degrees of freedom at the gear pair are θ_{wh} and θ_{pn} . They are defined as the rotation of the wheel and pinion about the first axis of their respective local coordinate system.

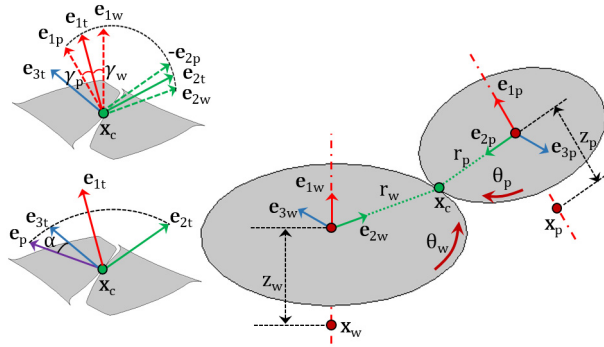


Figure 3-13: Sketch of a gear pair where the line of action is computed by the counterclockwise rotation of the tangent.

The rest of this section discusses the following topics:

- Gear Pair Compatibility Criteria
- Gear Ratio
- Gear Local Coordinate System
- Contact Point Position and Offset
- Gear Tooth Coordinate System
- Gear Pair Constraints
- Line of Action
- Contact Forces and Moments

GEAR PAIR COMPATIBILITY CRITERIA

In a **Gear Pair** node, you can select any two gears defined for the model. However, for the correct tooth meshing, a set of gears must fulfill the compatibility criteria.

Normal Module

The normal module of both gears must be the same.

$$m_{n, wh} = m_{n, pn}$$

Pressure Angle

The pressure angle of both gears must be the same.

$$\alpha_{wh} = \alpha_{pn}$$

Configuration

For parallel axis helical gears, the sum of the helix angles must be zero.

$$\beta_{wh} + \beta_{pn} = 0$$

For crossed axis helical gears, the sum of the helix angle must be nonzero.

$$\beta_{wh} + \beta_{pn} \neq 0$$

GEAR LOCAL COORDINATE SYSTEM

The local coordinate system for each gear is created using the gear axis and the center of rotation of both gears. This local coordinate system is attached to the gear; however, it does not rotate with the gear local rotation.

Initial Local Coordinate System

The first axis for the wheel ($\mathbf{e}_{10, wh}$) and pinion ($\mathbf{e}_{10, pn}$) is an input on the respective nodes. In 2D, it is assumed to be the out-of-plane direction.

For a parallel or intersecting configuration, the second axis is defined as:

$$\begin{aligned}\mathbf{e}_{20, wh} &= \mathbf{e}_{0, cd} - (\mathbf{e}_{0, cd} \cdot \mathbf{e}_{10, wh})\mathbf{e}_{10, wh} \\ \mathbf{e}_{20, pn} &= -(\mathbf{e}_{0, cd} - (\mathbf{e}_{0, cd} \cdot \mathbf{e}_{10, pn})\mathbf{e}_{10, pn})\end{aligned}$$

$$\mathbf{e}_{0, cd} = \frac{\mathbf{x}_{c, pn} - \mathbf{x}_{c, wh}}{\|\mathbf{x}_{c, pn} - \mathbf{x}_{c, wh}\|}$$

where $\mathbf{x}_{c, wh}$ and $\mathbf{x}_{c, pn}$ are the center of rotation of the wheel and pinion, respectively.

For a configuration that is neither parallel nor intersecting, the second axis is defined as:

$$\begin{aligned}\mathbf{e}_{20, wh} &= \mathbf{e}_{20} \operatorname{sgn}(\mathbf{e}_{20} \cdot \mathbf{e}_{0, cd}) \\ \mathbf{e}_{20, pn} &= -\mathbf{e}_{20, wh}\end{aligned}$$

$$\mathbf{e}_{20} = \mathbf{e}_{10, wh} \times \mathbf{e}_{10, pn}$$

In 2D:

$$\begin{aligned}\mathbf{e}_{20, wh} &= \mathbf{e}_{0, cd} \\ \mathbf{e}_{20, pn} &= -\mathbf{e}_{0, cd}\end{aligned}$$

The third axis is defined as follows:

$$\begin{aligned}\mathbf{e}_{30, wh} &= \mathbf{e}_{10, wh} \times \mathbf{e}_{20, wh} \\ \mathbf{e}_{30, pn} &= \mathbf{e}_{10, pn} \times \mathbf{e}_{20, pn}\end{aligned}$$

Local Coordinate System

The first axis is defined as follows:

$$\begin{aligned}\mathbf{e}_{1, wh} &= R_{wh} \cdot \mathbf{e}_{10, wh} \\ \mathbf{e}_{1, pn} &= R_{pn} \cdot \mathbf{e}_{10, pn}\end{aligned}$$

The second axis is defined as follows:

$$\begin{aligned}\mathbf{e}_{2, wh} &= (R_{wh} \cdot \mathbf{e}_{20, wh}) \cos \theta_{wh} - (R_{wh} \cdot \mathbf{e}_{30, wh}) \sin \theta_{wh} \\ \mathbf{e}_{2, pn} &= (R_{pn} \cdot \mathbf{e}_{20, pn}) \cos \theta_{pn} - (R_{pn} \cdot \mathbf{e}_{30, pn}) \sin \theta_{pn}\end{aligned}$$

The third axis is defined as follows:

$$\begin{aligned}\mathbf{e}_{2, wh} &= (R_{wh} \cdot \mathbf{e}_{20, wh}) \sin \theta_{wh} + (R_{wh} \cdot \mathbf{e}_{30, wh}) \cos \theta_{wh} \\ \mathbf{e}_{2, pn} &= (R_{pn} \cdot \mathbf{e}_{20, pn}) \sin \theta_{pn} + (R_{pn} \cdot \mathbf{e}_{30, pn}) \cos \theta_{pn}\end{aligned}$$

where R_{wh} and R_{pn} are the rotation matrices of the wheel and the pinion, respectively. The angles θ_{wh} and θ_{pn} are the rotations of the wheel and pinion about the first axis of their respective local coordinate system.

GEAR TOOTH COORDINATE SYSTEM

The gear tooth coordinate system is defined by rotating the gear local coordinate system with the helix angle and cone angle.

$$\begin{aligned}\mathbf{e}_{it, wh} &= T_{wh}^T \cdot \mathbf{e}_{i, wh} & i = 1, 2, 3 \\ \mathbf{e}_{it, pn} &= T_{pn}^T \cdot \mathbf{e}_{i, pn} & i = 1, 2, 3\end{aligned}$$

where T_{wh} and T_{pn} are the tooth transformation matrix for the wheel and pinion, respectively.

The gear-to-tooth transformation matrix is defined as:

$$T = \begin{bmatrix} \cos\gamma\cos\beta & -\sin\gamma & \cos\gamma\sin\beta \\ \sin\gamma\cos\beta & \cos\gamma & \sin\gamma\sin\beta \\ -\sin\beta & 0 & \cos\beta \end{bmatrix}$$



For the pinion tooth coordinate system, the sign of the second axis ($\mathbf{e}_{2t, pn}$) and third axis ($\mathbf{e}_{3t, pn}$) are reversed to match the directions with the wheel tooth coordinate system.

LINE OF ACTION

The line of action is the normal direction of the gear tooth surface at the contact point on the pitch circle. This is the direction along which the motion is transferred from one gear to another gear. It is defined by rotating the third axis of the tooth coordinate system (\mathbf{e}_{3t}) about the first axis of the tooth coordinate system (\mathbf{e}_{1t}) with the pressure angle (α).

The line of action, also known as the pressure angle direction, is defined as follows:

For clockwise rotation of the third axis:

$$\mathbf{e}_{pa} = \mathbf{e}_{2t, wh} \sin\alpha + \mathbf{e}_{3t, wh} \cos\alpha$$

For counterclockwise rotation of the third axis:

$$\mathbf{e}_{pa} = -\mathbf{e}_{2t, wh} \sin\alpha + \mathbf{e}_{3t, wh} \cos\alpha$$

GEAR RATIO

The gear ratio (g_r) of a gear pair is defined as the ratio of angular velocities of the wheel (ω_{wh}) and the pinion (ω_{pn}):

$$g_r = \frac{\omega_{wh}}{\omega_{pn}}$$

It can also be written as the ratio of number of teeth of the pinion (n_{pn}) and the wheel (n_{wh}):

$$g_r = \frac{n_{pn}}{n_{wh}}$$

CONTACT POINT POSITION AND OFFSET

Contact Point Position

The point of contact (\mathbf{x}_{cp}) on a gear in its local coordinate system can be defined as:

$$\begin{aligned}\mathbf{x}_{cp, wh} &= \mathbf{x}_{c, wh} + \mathbf{u}_{wh} + z_{wh} \mathbf{e}_{1, wh} + (r_{wh} - z_{wh} \tan \gamma_{wh}) \mathbf{e}_{2, wh} \\ \mathbf{x}_{cp, pn} &= \mathbf{x}_{c, pn} + \mathbf{u}_{pn} + z_{pn} \mathbf{e}_{1, pn} + (r_{pn} - z_{pn} \tan \gamma_{pn}) \mathbf{e}_{2, pn}\end{aligned}$$

where, for the wheel and pinion, respectively:

- $\mathbf{x}_{cp, wh}$ and $\mathbf{x}_{cp, pn}$ are the positions of the contact points
- $\mathbf{x}_{c, wh}$ and $\mathbf{x}_{c, pn}$ are the centers of rotation
- \mathbf{u}_{wh} and \mathbf{u}_{pn} are the displacement vectors at the center of rotation
- z_{wh} and z_{pn} are the contact point offsets from the center of rotation in the axial direction
- r_{wh} and r_{pn} are the pitch radii
- γ_{wh} and γ_{pn} are the cone angles

In 2D, the expressions for the position of contact points reduce to:

$$\begin{aligned}\mathbf{x}_{cp, wh} &= \mathbf{x}_{c, wh} + \mathbf{u}_{wh} + r_{wh} \mathbf{e}_{2, wh} \\ \mathbf{x}_{cp, pn} &= \mathbf{x}_{c, pn} + \mathbf{u}_{pn} + r_{pn} \mathbf{e}_{2, pn}\end{aligned}$$

Contact Point Offset

The contact point offset from the wheel or pinion center of rotation in the axial direction is defined as follows:

For a parallel or intersecting configuration, the contact point offset from the pinion center (z_{pn}) is the input. The contact point offset from the wheel center (z_{wh}) is defined as:

$$z_{wh} = (\mathbf{x}_{c, pn} - \mathbf{x}_{c, wh} + z_{pn} \mathbf{e}_{10, pn} + (r_{pn} - z_{pn} \tan \gamma_{pn}) \mathbf{e}_{20, pn}) \cdot \mathbf{e}_{10, wh}$$

For a configuration that is neither parallel nor intersecting, the contact point offset from the pinion center (z_{pn}) and the contact point offset from the wheel center (z_{wh}) are defined as:

$$z_{pn} = \frac{-(\mathbf{x}_{cd} \cdot \mathbf{e}_{10, pn}) - (\mathbf{x}_{cd} \cdot \mathbf{e}_{10, wh}) x_{wh, pn}}{1 - x_{wh, pn}^2}$$

$$z_{wh} = \frac{(\mathbf{x}_{cd} \cdot \mathbf{e}_{10, wh}) - (\mathbf{x}_{cd} \cdot \mathbf{e}_{10, pn})x_{wh, pn}}{1 - x_{wh, pn}^2}$$

where

$$\begin{aligned}\mathbf{x}_{cd} &= \mathbf{x}_{c, pn} - \mathbf{x}_{c, wh} \\ x_{wh, pn} &= \mathbf{e}_{10, wh} \cdot \mathbf{e}_{10, pn}\end{aligned}$$

GEAR PAIR CONSTRAINTS

Rotation Constraint

This constraint relates the pinion rotation to the wheel rotation:

$$\theta_{pn} = -\frac{\theta_{wh}}{g_r} + \theta_{el} + \theta_{et} + \theta_{bl}$$

where θ_{el} , θ_{et} , and θ_{bl} are the transmission error (elasticity), transmission error (static), and transmission error (backlash), respectively.

By default, there is no elasticity, transmission error, or backlash in a gear pair hence all these transmission errors are zero.



For more details about these transmission errors, see the [Theory for Gear Elasticity](#), [Theory for Transmission Error](#), and [Theory for Backlash](#) sections.

Point Contact Constraint

At the contact point, the relative motion along the line of action must be zero. This can be written as:

$$(\mathbf{x}_{cp, pn} - \mathbf{x}_{cp, wh}) \cdot \mathbf{e}_{pa} = 0$$

Line Contact Constraint

In case of a line contact model, an additional constraint is added at the second contact point. This can be written as:

$$\left(\mathbf{x}_{cp2, pn} - \mathbf{x}_{cp2, wh} - \frac{w_{pn}}{2} \theta_{cl} \mathbf{e}_{3, pn} \right) \cdot \mathbf{e}_{pa} = 0$$

where $\mathbf{x}_{cp2,wh}$ and $\mathbf{x}_{cp2,pn}$ are the position of the second contact point and can be defined as:

$$\left(\mathbf{x}_{cp2,wh} = \text{subst}\left(x_{cp,wh}, z_{wh}, z_{wh} + \frac{w_{pn} \cos \gamma_{wh}}{2 \cos \gamma_{pn}}\right) \right)$$

$$\mathbf{x}_{cp2,pn} = \text{subst}\left(x_{cp,pn}, z_{pn}, z_{pn} + \frac{w_{pn}}{2}\right)$$

Here, w_{pn} and θ_{cl} are the working width of the pinion and the relative rotation about center line, respectively. By default, there is no elasticity on a gear pair hence the relative rotation about the center line is zero.

Unique Triad Constraint

In case of planetary gears, there is a constraint to ensure that the local coordinate system of both the gears are still unique and well defined. This can be written as:

$$\mathbf{e}_{2t,wh} \cdot \mathbf{e}_{3t,pn} = 0$$

CONTACT FORCES AND MOMENTS

Contact Force

When you compute a contact force using weak constraints, the point contact constraint is implemented in the weak form and the corresponding Lagrange multiplier gives the contact force (\mathbf{F}_c).

Similarly, when you compute a contact force using a penalty method, the point contact constraint is implemented using the penalty factor and the corresponding penalty force gives the contact force (\mathbf{F}_c).

For the case of a line contact model, the second contact force (\mathbf{F}_{c2}) is computed by implementing the line contact constraint in the weak form or by using the penalty factor.

Contact Moment

When you compute a contact force using weak constraints, the unique triad constraint is implemented in the weak form and the corresponding Lagrange multiplier gives the moment at the contact point (\mathbf{M}_c).

Similarly, when you compute a contact force computation using a penalty method, the unique triad constraint is implemented using the penalty factor and the corresponding penalty moment gives the moment at the contact point (\mathbf{M}_c).

Force and Moment at Gear Centers

The forces at the gear centers are defined as:

$$\begin{aligned}\mathbf{F}_{wh} &= -F_c \mathbf{e}_{pa} \\ \mathbf{F}_{pn} &= F_c \mathbf{e}_{pa}\end{aligned}$$

The moments at the gear centers are defined as:

$$\begin{aligned}\mathbf{M}_{wh} &= (\mathbf{x}_{cp, pn} - \mathbf{x}_{wh} - \mathbf{u}_{wh}) \times (-F_c \mathbf{e}_{pa}) + M_c (\mathbf{e}_{2t, wh} \times \mathbf{e}_{3t, pn}) \\ \mathbf{M}_{pn} &= (\mathbf{x}_{cp, pn} - \mathbf{x}_{pn} - \mathbf{u}_{pn}) \times (F_c \mathbf{e}_{pa}) + (-M_c) (\mathbf{e}_{2t, wh} \times \mathbf{e}_{3t, pn})\end{aligned}$$

For a line contact model, additional contributions from the contact force F_{c2} are added to the forces and moments at the gear centers. These contributions can be defined as:

$$\begin{aligned}\mathbf{F}_{wh} &= -F_{c2} \mathbf{e}_{pa} \\ \mathbf{F}_{pn} &= F_{c2} \mathbf{e}_{pa} \\ \mathbf{M}_{wh} &= (\mathbf{x}_{cp2, pn} - \mathbf{x}_{wh} - \mathbf{u}_{wh}) \times (-F_{c2} \mathbf{e}_{pa}) \\ \mathbf{M}_{pn} &= (\mathbf{x}_{cp2, pn} - \mathbf{x}_{pn} - \mathbf{u}_{pn}) \times (F_{c2} \mathbf{e}_{pa})\end{aligned}$$

Theory for Worm and Wheel

The worm and wheel node connects a worm gear with a spur or helical gear in such a way that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of both the gears are independent of each other.

When friction is included on a worm and wheel node, frictional forces in the plane perpendicular to the line of action are added on both the gears.

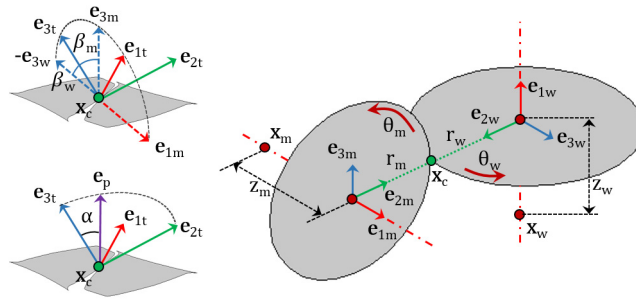


Figure 3-14: Sketch of a worm and wheel pair where line of action is computed by the clockwise rotation of the tangent.

The degrees of freedom at the worm and wheel are θ_{wm} and θ_{wh} . They are defined as the rotation of the worm and wheel about the first axis of their respective local coordinate system.

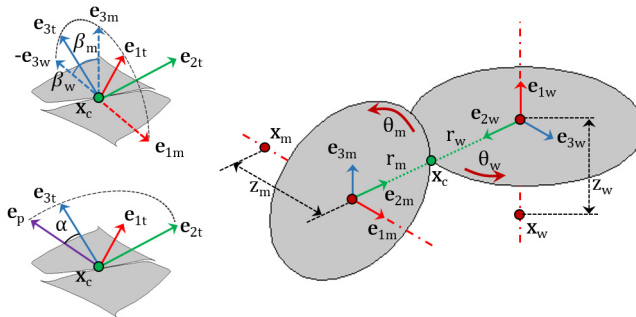


Figure 3-15: The sketch of a worm and wheel pair where line of action is computed by the counterclockwise rotation of the tangent.



In the worm and wheel node, the configuration is always neither parallel nor intersecting and the contact model is point contact.

The rest of the theory for the worm and wheel node is similar to the [Theory for Gear Pairs](#).



The gear pair node connects a wheel with pinion whereas the worm and wheel node connects a worm with wheel. To get worm and wheel node specific equations, replace wheel with worm and pinion with wheel everywhere.

Theory for Rack and Pinion

The rack and pinion node connects a spur or helical rack with a spur or helical gear in such a way that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of the rack and gear are independent of each other.

For a line contact model, one additional constraint is added to restrict the relative rotation about a line joining the rack and gear centers. When friction is included on a rack and pinion node, frictional forces in the plane perpendicular to the line of action are added on the rack and pinion.

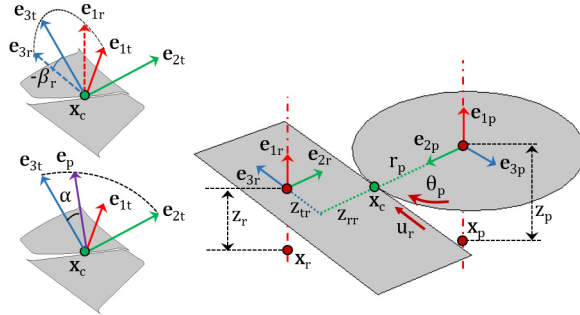


Figure 3-16: The sketch of a rack and pinion pair where line of action is computed by the clockwise rotation of the tangent.

The degrees of freedom at the rack and pinion are u_{rk} and θ_{pn} . They are defined as the displacement of rack along the third axis and the rotation of pinion about the first axis of their respective local coordinate system, respectively.

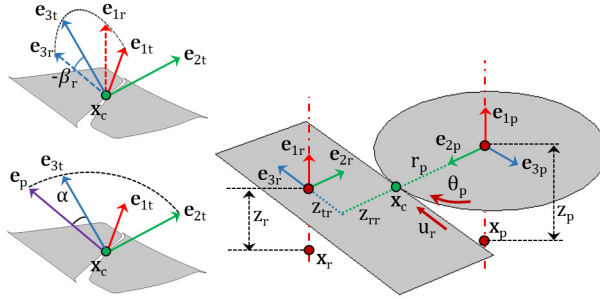


Figure 3-17: The sketch of a rack and pinion pair where line of action is computed by the counterclockwise rotation of the tangent.



The rest of the theory for the rack and pinion node is similar to the [Theory for Gear Pairs](#) with the following changes.

The gear pair node connects a wheel with pinion whereas the rack and pinion node connects a rack with pinion. To get rack and pinion node specific equations, replace wheel with rack everywhere.

GEAR PAIR COMPATIBILITY CRITERIA

Normal Pitch

The normal pitch of the rack and pinion must be same.

$$P_{n, rk} = \pi m_{n, pn}$$



For more details, refer the [Gear Pair Compatibility Criteria](#) for the gear pair node.

GEAR LOCAL COORDINATE SYSTEM

Initial Local Coordinate System

The third axis for the rack ($\mathbf{e}_{30, rk}$) and the first axis of the pinion ($\mathbf{e}_{10, pn}$) is the input on the respective nodes. In 2D, the first axis of the rack and pinion are assumed to be the out-of-plane direction.

The second axis is defined as follows:

$$\mathbf{e}_{20, rk} = \mathbf{e}_{20} \operatorname{sgn}(\mathbf{e}_{20} \cdot \mathbf{e}_{0, cd})$$

$$\mathbf{e}_{20, pn} = -\mathbf{e}_{20, rk}$$

$$\mathbf{e}_{20} = \mathbf{e}_{30, rk} \times \mathbf{e}_{10, pn}$$

$$\mathbf{e}_{0, cd} = \frac{\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk}}{\|\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk}\|}$$

The first axis of rack and the third axis of pinion are defined as follows:

$$\mathbf{e}_{10, rk} = \mathbf{e}_{20, rk} \times \mathbf{e}_{30, rk}$$

$$\mathbf{e}_{30, pn} = \mathbf{e}_{10, pn} \times \mathbf{e}_{20, pn}$$



For more details, refer the [Gear Local Coordinate System](#) for the gear pair node.

CONTACT POINT POSITION AND OFFSET

Contact Point Position

The point of contact (\mathbf{x}_{cp}) on a rack in its local coordinate system can be defined as:

$$\mathbf{x}_{cp, rk} = \mathbf{x}_{c, rk} + \mathbf{u}_{rk} + z_{rk} \mathbf{e}_{1, rk} + z_{r, rk} \mathbf{e}_{2, rk} + (z_{t, rk} - u_{rk}) \mathbf{e}_{3, rk}$$

where $z_{r, rk}$ and $z_{t, rk}$ are the contact point offset from the center of rotation in the radial and tangential direction, respectively.

In 2D, the expression for the position of contact point reduces to:

$$\mathbf{x}_{cp, rk} = \mathbf{x}_{c, rk} + \mathbf{u}_{rk} + z_{r, rk} \mathbf{e}_{2, rk} + (z_{t, rk} - u_{rk}) \mathbf{e}_{3, rk}$$

Contact Point Offset

The contact point offset from the pinion center (z_{pn}) is the input.

The contact point offset from the rack center in the axial direction (z_{rk}) is defined as:

$$z_{rk} = (\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk} + z_{pn} \mathbf{e}_{10, pn}) \cdot \mathbf{e}_{10, rk}$$

The contact point offset from the rack center in the radial direction ($z_{r,rk}$) is defined as:

$$z_{r, rk} = (\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk}) \cdot \mathbf{e}_{20, rk} - r_{pn}$$

The contact point offset from the rack center in the tangential direction ($z_{t,rk}$) is defined as:

$$z_{t, rk} = (\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk} + z_{pn} \mathbf{e}_{10, pn}) \cdot \mathbf{e}_{30, rk}$$

In 2D, the expression reduces to:

$$z_{t, rk} = (\mathbf{x}_{c, pn} - \mathbf{x}_{c, rk}) \cdot \mathbf{e}_{30, rk}$$



For more details, refer the [Contact Point Position and Offset](#) for the gear pair node.

GEAR PAIR CONSTRAINTS

Translation-Rotation constraint

This constraint relates the pinion rotation with the rack displacement:

$$\theta_{pn} = -\frac{u_{rk} \cos \beta_{rk}}{r_{pn} \cos \beta_{pn}} + \theta_{el} + \theta_{et} + \theta_{bl}$$



For more details, refer the [Gear Pair Constraints](#) for the gear pair node.

Theory for Gear Elasticity

The **Gear Elasticity** node can optionally be added on gear pairs. In this node, you can specify the elastic properties of the gear mesh, such as mesh stiffness, mesh damping, and contact ratio. By default, the gear mesh is assumed rigid.

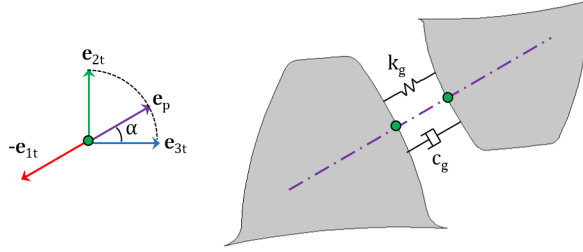


Figure 3-18: A sketch of a pair of teeth, showing the point of contact and the direction in which mesh stiffness and damping are interpreted.

The rest of this section discusses the following topics:

- Degrees of Freedom
- Mesh Stiffness
- Elastic Moments
- Mesh Damping

DEGREES OF FREEDOM

In this node, the transmission error (elasticity), θ_{el} , is a degree of freedom. This transmission error is added in the rotation constraint of the gear pair.

In the case of a line contact model, one more degree of freedom, θ_{cl} , is added. This is the relative rotation about the line joining the centers of two gears. The contribution of this rotation is added in the line contact constraint of the gear pair.

ELASTIC MOMENTS

The elastic moment about the first axis of the tooth coordinate system and its virtual work contribution are defined as:

$$M_{el} = -(k_g \theta_{el} + c_g \dot{\theta}_{el}) (\mathbf{e}_{pa} \cdot \mathbf{e}_{3, pn})^2 r_{pn}^2$$

$$\delta W = M_{el} \cdot \delta \theta_{el}$$

In the case of a line contact model, the elastic moment about the second axis of the tooth coordinate system and its virtual work contribution are defined as:

$$M_{cl} = -(k_g \theta_{cl} + c_g \dot{\theta}_{cl}) (\mathbf{e}_{pa} \cdot \mathbf{e}_{3, pn})^2 \frac{w_{pn}^2}{12}$$

$$\delta W = M_{cl} \cdot \delta \theta_{cl}$$

where

- k_g is the mesh stiffness of gear pair
- c_g is the mesh damping of gear pair
- r_{pn} is the pitch radius of pinion
- w_{pn} is the working width of pinion



The mesh damping contribution to the elastic moments will be zero in the case of a stationary analysis.

MESH STIFFNESS



The mesh stiffness expressions in this section are derived for a **Gear Pair** node.

To get equations that are specific to the **Worm and Wheel** node, replace “wheel” with “worm” and “pinion” with “wheel” everywhere.

To get equations that are specific to the **Rack and Pinion** node, replace “wheel” with “rack” everywhere.

Tooth Stiffness of Individual Gears

The mesh stiffness of the gear pair is defined as:

$$k_g = \frac{k_{wh}k_{pn}}{k_{wh} + k_{pn}}$$

where k_{wh} and k_{pn} are the mesh stiffness of the wheel and pinion, respectively.

The mesh stiffness of the wheel, for different contact ratios, is defined as follows.



The mesh stiffness of the pinion can be derived in a similar manner.

Contact ratio is constant and set to 1:

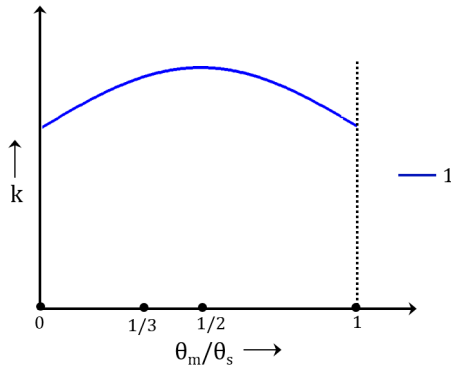


Figure 3-19: The graph shows the variation of the gear tooth stiffness in a mesh cycle.

The mesh stiffness of the wheel is defined as:

$$k_{wh} = k_{t, wh}(\theta_{m, wh})$$

where

- $k_{t, wh}$ is the gear tooth stiffness as function of mesh cycle
- $\theta_{m, wh}$ is the mesh cycle

The mesh cycle for the wheel is defined as:

$$\theta_{m, wh} = \text{mod}(\theta_{wh}, \theta_{s, wh}) \quad \theta_{s, wh} = \frac{2\pi}{n_{wh}}$$



The mesh cycle for the rack is different from other gear types and can be defined as:

$$u_{m, rk} = \text{mod}(u_{rk}, p_{rk})$$

Contact ratio is constant and set to 2:

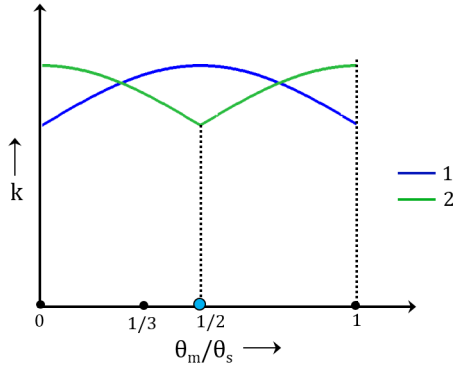


Figure 3-20: The graph shows the variation of the gear tooth stiffness of the first and second teeth in a mesh cycle.

The mesh stiffness of the wheel is defined as:

$$k_{wh} = k_{t,wh}(\theta_{m,wh}) + k_{t2,wh}$$

where $k_{t2,wh}$ is the tooth stiffness of the second tooth. It is defined as:

$$k_{t2,wh} = k_{t,wh}\left(\theta_{m,wh} + \frac{\theta_{s,wh}}{2}\right)$$

Contact ratio is constant and set to 3:

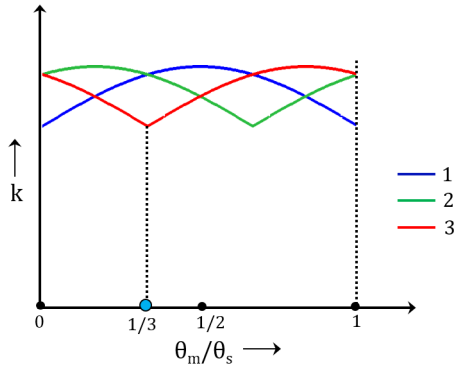


Figure 3-21: The graph shows the variation of the gear tooth stiffness of the first, second, and third teeth in a mesh cycle.

The mesh stiffness of the wheel is defined as:

$$k_{wh} = k_{t,wh}(\theta_{m,wh}) + k_{t2,wh} + k_{t3,wh}$$

where $k_{t2,wh}$ and $k_{t3,wh}$ are the tooth stiffness of the second and third teeth, respectively. They are defined as:

$$k_{t2,wh} = k_{t,wh}\left(\theta_{m,wh} + \frac{\theta_{s,wh}}{3}\right)$$

$$k_{t3,wh} = k_{t,wh}\left(\theta_{m,wh} + \frac{2\theta_{s,wh}}{3}\right)$$

Contact ratio is varying and maximum contact ratio is set to 2:

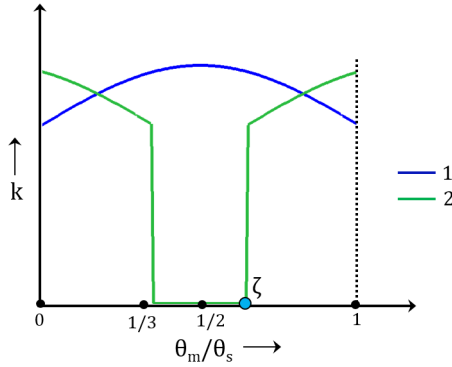


Figure 3-22: The graph shows the variation of the gear tooth stiffness of the first and second teeth in a mesh cycle.

The mesh stiffness of the wheel is defined as:

$$k_{wh} = k_{t,wh}(\theta_{m,wh}) + k_{t2f,wh} + k_{t2b,wh}$$

where $k_{t2f,wh}$ and $k_{t2b,wh}$ are the tooth stiffness of the second forward and backward teeth. They are defined as:

$$k_{t2f,wh} = (\theta_{m,wh} < (1 - \zeta)\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} + \zeta\theta_{s,wh})$$

$$k_{t2b,wh} = (\theta_{m,wh} \geq \zeta\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} - \zeta\theta_{s,wh})$$

where ζ is the next tooth engagement position in the mesh cycle. The value of ζ must be in the following range:

$$\frac{1}{2} \leq \zeta \leq 1$$

Contact ratio is varying and maximum contact ratio is set to 3:

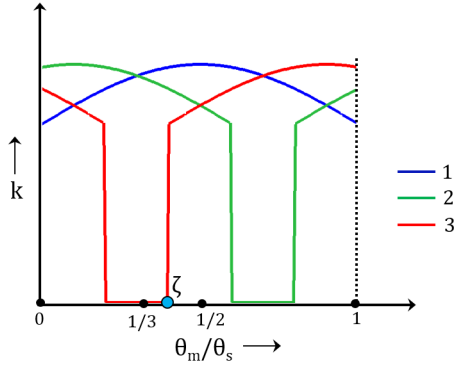


Figure 3-23: The graph shows the variation of the gear tooth stiffness of the first, second, and third teeth in a mesh cycle.

The mesh stiffness of the wheel is defined as:

$$k_{wh} = k_{t,wh}(\theta_{m,wh}) + k_{t2f,wh} + k_{t2b,wh} + k_{t3f,wh} + k_{t3b,wh}$$

where

- $k_{t2f,wh}$ and $k_{t2b,wh}$ are the tooth stiffness of the second forward and backward teeth
- $k_{t3f,wh}$ and $k_{t3b,wh}$ are the tooth stiffness of the third forward and backward teeth

$$\begin{aligned} k_{t2f,wh} &= (\theta_{m,wh} < (1 - \zeta)\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} + \zeta\theta_{s,wh}) \\ k_{t2b,wh} &= (\theta_{m,wh} \geq 2\zeta\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} - 2\zeta\theta_{s,wh}) \\ k_{t3f,wh} &= (\theta_{m,wh} < (1 - 2\zeta)\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} + 2\zeta\theta_{s,wh}) \\ k_{t3b,wh} &= (\theta_{m,wh} \geq \zeta\theta_{s,wh}) \cdot k_{t,wh}(\theta_{m,wh} - \zeta\theta_{s,wh}) \end{aligned}$$

The value of ζ must be in the following range:

$$\frac{1}{3} \leq \zeta \leq \frac{1}{2}$$

Total Stiffness of Gear Pair

The mesh stiffness of gear pair is defined as follows:

Function of wheel mesh cycle:

$$k_g = k_g(\theta_{m, wh})$$
$$\theta_{m, wh} = \text{mod}(\theta_{wh}, \theta_{s, wh}) \quad \theta_{s, wh} = \frac{2\pi}{n_{wh}}$$

Function of wheel full revolution:

$$k_g = k_g(\theta_{f, wh})$$
$$\theta_{f, wh} = \text{mod}(\theta_{wh}, 2\pi)$$

Function of pinion mesh cycle:

$$k_g = k_g(\theta_{m, pn})$$
$$\theta_{m, pn} = \text{mod}(\theta_{pn}, \theta_{s, pn}) \quad \theta_{s, pn} = \frac{2\pi}{n_{pn}}$$

Function of pinion full revolution:

$$k_g = k_g(\theta_{f, pn})$$
$$\theta_{f, pn} = \text{mod}(\theta_{pn}, 2\pi)$$

MESH DAMPING

The mesh damping of a gear pair can be a constant value or it can be written as a function of mesh cycle or mesh stiffness.

Theory for Transmission Error

The **transmission Error** node can optionally be added on gear pairs. In this node, the static transmission error can be specified and added in the rotation constraint of the gear pair. By default, there is no transmission on the gear pair and it is assumed to be an ideal pair.

The static transmission error could be due to geometrical errors and geometrical modifications, such as gear runouts, misalignments, tooth tip, and root relief.

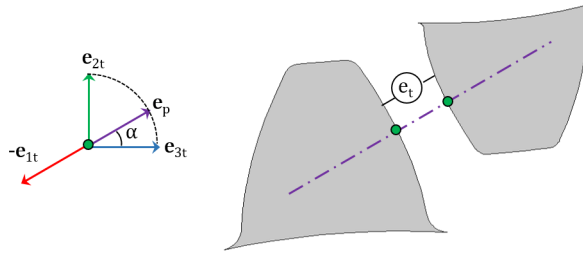


Figure 3-24: Sketch of a pair of teeth, showing the point of contact and the direction in which the transmission error is interpreted.



The transmission error expression in this section is derived for a Gear Pair node.

To get equations specific to the Worm and Wheel node, replace “wheel” with “worm” and “pinion” with “wheel” everywhere.

To get equations specific to the Rack and Pinion node, replace “wheel” with “rack” everywhere.

The static transmission error can be defined as:

$$\theta_{et} = \frac{e_t}{r_{pn}} \|(\mathbf{e}_{pa} \cdot \mathbf{e}_{3, pn})\|$$

$$e_t = e_{wh}(\theta_m, wh) + e_{pn}(\theta_m, pn)$$

where e_{wh} and e_{pn} are the wheel and pinion transmission errors, respectively.

In the above equation, e_{wh} and e_{pn} are written as a function of respective mesh cycle. However, it is also possible to write them individually as a function of a full revolution.

Similarly, it is also possible to directly specify e_t as a function of the mesh cycle and a full revolution of a wheel or pinion.

Theory for Backlash

The **Backlash** node can optionally be added on gear pairs. The transmission error due to backlash is added in the rotation constraint of the gear pair. By default, there is no backlash on a gear pair.

The backlash on a gear pair plays an important role for unloaded gears. For loaded gears, it does not affect the dynamics of gears to a large extent.



The backlash is taken into account only in the case of a time-dependent analysis.

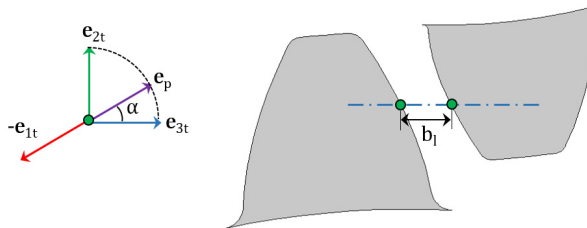


Figure 3-25: Sketch of a pair of teeth, showing the point of contact and the direction in which backlash is interpreted

The transmission error (backlash), θ_{bl} , is degree of freedom in this node. The backlash moment about the first axis of the tooth coordinate system and its virtual work contribution are defined as:

$$M_b = -p_{bm}((\theta_{bl} < 0)\theta_{bl} + (\theta_{bl} > \theta_b)(\theta_{bl} - \theta_b))$$

$$\delta W = M_b \cdot \delta\theta_{bl}$$

$$\theta_b = \frac{\|b_1\|}{r_{pn}}$$

where b_1 and p_{bm} are the backlash and modified penalty factor, respectively.

To prevent the solver from taking larger time steps, the penalty factor is modified in the following manner:

$$p_{bm} = p_b + \text{if}(((\theta_{bl} < 0) + (\theta_{bl} > \theta_b)) \cdot (\Delta t > \Delta t_{\max}), \infty, 0)$$



The penalty factor is not modified for quasi-static analysis or when the manual control on time stepping is deselected.

Theory for Gear Friction

The **Friction** node can optionally be added on gear pairs. In cases where friction is included on a gear pair, frictional forces in the plane perpendicular to the line of action are added on both gears.

By default, there is no friction on **Gear Pair** and **Rack and Pinion**. It is, however, included for **Worm and Wheel**. The worm and wheel pair has always neither a parallel nor an intersecting configuration and in this configuration there is always a slip even under perfect conditions.



Friction can be applied to gear pairs only in time-dependent analysis.

FRICITION MODEL

The friction force is modeled using a continuous friction law, which is capable of modeling sliding-sticking phenomena. A strict application of Coulomb's law involves discrete transition from sticking to sliding and vice versa, as dictated by a vanishing relative velocity. These discrete transitions cause numerical difficulties and to avoid them, the friction force is approximated with a continuous friction law:

$$F_f = -\mu N \frac{v}{|v|} \left(1 - e^{\frac{-|v|}{v_0}} \right)$$

where F_f is the friction force, μ is the frictional coefficient, N is the normal force, v is the slip velocity, and v_0 is the characteristic slip velocity. The term

$$\left(1 - e^{\frac{-|v|}{v_0}} \right)$$

is called the regularization factor.

The regularization factor smooths the friction force discontinuity. The characteristic slip velocity should be made small in comparison to the characteristic relative velocities encountered during the simulation. The continuous friction law describes both sliding and sticking behavior; that is, it completely replaces Coulomb's law. Sticking is replaced by creeping between the contacting bodies with a small relative velocity.

NORMAL FORCE

The normal force at the contact point can be taken as the computed contact force. Alternatively, it can also be entered explicitly.

SLIP AT CONTACT POINT

The slip vector at the contact point of a gear pair is defined as:

$$\begin{aligned}\mathbf{u}_s &= (\mathbf{x}_{cp, pn} - \mathbf{x}_{cp, wh}) + (r_{pn}\theta_{pn}\mathbf{e}'_{3, pn} - r_{wh}\theta_{wh}\mathbf{e}'_{3, wh}) \\ \mathbf{e}'_{3, wh} &= \mathbf{e}_{3, wh} - (\mathbf{e}_{3, wh} \cdot \mathbf{e}_{pa})\mathbf{e}_{pa} \\ \mathbf{e}'_{3, pn} &= \mathbf{e}_{3, pn} - (\mathbf{e}_{3, pn} \cdot \mathbf{e}_{pa})\mathbf{e}_{pa}\end{aligned}$$

A similar expression can be written for a worm and wheel pair. However, for a rack and pinion pair, the definition is slightly different and the slip vector is defined as:

$$\mathbf{u}_s = (\mathbf{x}_{cp, pn} - \mathbf{x}_{cp, rk}) + (r_{pn}\theta_{pn}\mathbf{e}'_{3, pn} - u_{rk}\mathbf{e}'_{3, rk})$$

In 2D, the slip vector expression reduces to:

$$\mathbf{u}_s = (\mathbf{x}_{cp, pn} - \mathbf{x}_{cp, rk})$$

In the case of a line contact model, the slip at the second contact point is defined as:

$$\mathbf{u}_{s2} = (\mathbf{x}_{cp2, pn} - \mathbf{x}_{cp2, wh}) + (r_{pn}\theta_{pn}\mathbf{e}'_{3, pn} - r_{wh}\theta_{wh}\mathbf{e}'_{3, wh})$$

FRICTION FORCE

The friction force at the contact point and its virtual work contribution are defined as:

$$\begin{aligned}\mathbf{F}_f &= -\min(\mu N + F_{f, r}, F_{f, \max}) \frac{\dot{\mathbf{u}}_s}{\|\dot{\mathbf{u}}_s\|} \left(1 - e^{-\frac{\|\dot{\mathbf{u}}_s\|}{v_0}} \right) \\ \delta W &= \mathbf{F}_f \cdot \delta \mathbf{u}_s\end{aligned}$$

where $F_{f, r}$ and $F_{f, \max}$ are the additional sliding resistance and the maximum friction force, respectively.

Similarly, in the case of a line contact model, the friction force at the second contact point and its virtual work contribution is defined as:

$$\mathbf{F}_{f2} = -\min(\mu N_2 + F_{f,r}, F_{f,\max}) \frac{\dot{\mathbf{u}}_{s2}}{\|\dot{\mathbf{u}}_{s2}\|} \left(1 - e^{-\frac{\|\dot{\mathbf{u}}_{s2}\|}{v_0}} \right)$$

$$\delta W = \mathbf{F}_{f2} \cdot \delta \mathbf{u}_{s2}$$

ENERGY DISSIPATION RATE


The energy dissipation rate due to friction at the contact point is defined as:

$$Q_f = -\mathbf{F}_f \cdot \dot{\mathbf{u}}_s$$

For a line contact model, the additional contribution in the energy dissipation rate is defined as:

$$Q_f = -\mathbf{F}_{f2} \cdot \dot{\mathbf{u}}_{s2}$$



Multibody Dynamics

This chapter describes the Multibody Dynamics interface, which is found under the **Structural Mechanics** branch () when adding a physics interface.

In this chapter:

- [The Multibody Dynamics Interface](#)

The Multibody Dynamics Interface

The **Multibody Dynamics (mbd)** interface () found under the **Structural Mechanics** branch () when adding a physics interface, is intended for analysis of mechanical assemblies. The parts in the assembly can be rigid or flexible, and are connected by various types of joints, gears, springs, or dampers. Flexible parts can be defined using solid, shell, or beam elements. There are many types of joints, such as hinges or ball joints, which can be used based on the type of connection required between components. There are many types of gears, such as spur, helical, bevel, worm, or rack, which can be used for power transmission. Computed results are displacements, velocities, accelerations, joint forces, gear contact forces, and — in flexible parts — stresses. The joints can be given properties such as spring constants, damping, friction, and limits on movement. The gears pairs can also be given properties such as gear mesh stiffness, mesh damping, backlash, transmission error, and friction.

The Linear Elastic Material is the default material model. It adds the equations for the displacements in a linear elastic solid and has a Settings window to define the elastic and inertia properties of a material. As an alternative, a domain can be made into a rigid body, using the Rigid Domain material model. A rigid domain adds the rigid body dynamics equations and has a Settings window to define inertia properties of a material.

With the Nonlinear Structural Materials Module, you can also incorporate nonlinear material models like hyperelasticity or plasticity by adding a Solid Mechanics interface to the model.

When the **Multibody Dynamics** interface is added, these default nodes are also added to the **Model Builder** — **Linear Elastic Material**, **Free** (a boundary condition where boundaries are free, with no loads or constraints), and **Initial Values** (only applicable for flexible domains). Then, from the **Physics** toolbar, add features that implement other multibody dynamics properties. You can also right-click **Multibody Dynamics** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the **name** string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is **mbd**.

2D APPROXIMATION

From the **2D approximation** list select **Plane stress** or **Plane strain** (the default). Plane stress is relevant for structures which are thin in the out-of-plane direction, such as a thin plate. Plane strain is relevant when the 2D model can be considered as a cut through an object which is long in the out-of-plane direction. This information is needed if the components are flexible and modeled using **Linear Elastic Material**. For more information see the theory section.



When modeling using plane stress, the Solid Mechanics interface solves for the out-of-plane strain displacement derivative, $\frac{\partial w}{\partial Z}$, in addition to the displacement field **u**.

When combining Multibody Dynamics with other types of physics, there is often an assumption that the out-of-plane extension is infinitely long. This is the case in for example Acoustic-Structure interaction problems. In these cases, **Plane strain** is usually the correct choice.

THICKNESS

For 2D components, enter a value or expression for the **Thickness** *d*. The default value of 1 m is suitable for plane strain models, where it represents a unit-depth slice, for example. For plane stress models, enter the actual thickness, which should be small compared to the size of the plate for the plane stress assumption to be valid. In Acoustic-Structure Interaction problems, the **Thickness** should be set to 1 m.



Use a [Change Thickness](#) node to change thickness in parts of the geometry if necessary.

STRUCTURAL TRANSIENT BEHAVIOR

From the **Structural transient behavior** list, select **Include inertial terms** or **Quasi-static**. Use **Quasi-static** to treat the mechanical behavior as quasi-static (with no mass effects

— that is, no second-order time derivatives). Selecting this option gives a more efficient solution for problems where the variation in time is slow when compared to the natural frequencies of the system.

REFERENCE POINT FOR MOMENT COMPUTATION

Enter the coordinates for the **Reference point for moment computation** \mathbf{x}_{ref} (variable `refpnt`). The resulting moments (applied or as reactions) are then computed relative to this reference point. During the results and analysis stage, the coordinates can be changed in the **Parameters** section in the result nodes.

INITIAL VALUES

Specify initial values for a rigid body translation and rotation. The values specified here are, as a default, inherited by the initial values settings for **Rigid Domain** and **Linear Elastic Material**.


- **Center of rotation** \mathbf{X}_c
- **Displacement at center of rotation** \mathbf{u}
- **Velocity at center of rotation** $\partial\mathbf{u}/\partial t$
- *For 3D components only:* **Axis of rotation** Ω .
- **Angle of rotation** ϕ
- **Angular velocity** ω (3D components) and $\partial\phi/\partial t$ (2D components)

RESULTS

In this section you can modify the postprocessing defaults for the interface.

You can select a **Body defining reference frame** from the list of all rigid domains, attachments, and gears used in the model. It is used to visualize the relative motion of objects with respect to the selected body.

Variables `u_ref`, `v_ref` and `w_ref`, being the three components of the displacement field with respect to the reference body are available in the result menus. Also, the variables `mbd.disp_ref` and `mbd.vel_ref` contain the total displacement and velocity with respect to the reference body.

When you change the reference body, you need to update the solution before you can access the new definitions of these variables. To do that, right-click the **Study** node and select **Update Solution** () (or press F5).

JOINTS SUMMARY

This is an information table that displays the name of all the joints used in a model and their source and destination attachments. This table summarizes all the connections present in a model and can be used to review the connections.

RIGID BODY DOF SUMMARY

This is an information table that displays the number of degrees of freedom and constraints introduced by rigid domains and joints. You can use this information to determine whether your model has an appropriate number of constraints or not when running pure rigid body analysis.

Rigid Body DOF Summary				
	N	DOF	Prescribed	Constraints
Rigid bodies	14	98	0	20
Prismatic joints	3	3	3	18
Hinge joints	8	8	0	48
Cylindrical joints	4	8	2	24
Ball joints	2	8	0	14
Total	-	125	5	124
Net	-	0	-	4

The contents of the columns are:

- **N**: The number of features of a certain type.
- **DOF**: The number of degrees of freedom added to the model by all features of this type.
- **Prescribed**: The number of degrees of freedom controlled by **Prescribed Motion** and similar conditions for all features of this type.
- **Constraints**: The number of constraints for all features of this type. Both internal constraints and constraints added explicitly by for example **Fixed** conditions are included.

The last two rows of the table contain a summary. In the **Total** row, the number of DOFs, prescribed conditions, and constraints are summed.

The **Net** row contains the net number of degrees of freedom of the model — that is, the difference between all degrees of freedom and the constraints and prescribed motions. A negative net number of degrees of freedom indicates that the mechanism is overconstrained and is not shown. In that case, the net number of constraints is instead displayed.

Some constraints are conditional and are not present during the whole analysis. Such constraints are counted in the summary table, irrespective of their actual status.

The contents of the table do not change if you suppress any of the contributing features from the settings in the study step.

In general, and if there are flexible bodies in the system, the model's total number of degrees of freedom increase significantly. Such degrees of freedom are not accounted for in the summary.

DEPENDENT VARIABLES

The physics interface uses the global spatial components of the **Displacement field** u as dependent variables in the flexible domains. The default names for the components are (u, v, w) in 3D. In 2D the component names are (u, v) , and in 2D axial symmetry they are (u, w) . You can, however, not use the “missing” component name in the 2D cases as a parameter or variable name because it is still used internally.

You can change both the field name and the individual component names. If a new field name coincides with the name of another displacement field, the two fields (and the interfaces which define them) share degrees of freedom and dependent variable component names. You can use this behavior to connect a Multibody Dynamics interface to a Shell directly attached to the boundaries of the solid domain, or to a Solid Mechanics interface sharing a common boundary.

A new field name must not coincide with the name of a field of another type (that is, it must contain a displacement field), or with a component name belonging to some other field. Component names must be unique within a model except when two interfaces share a common field name.

Each rigid domain, attachment, and joint also adds a number of global dependent variables depending on the number of ODEs needed to represent its motion.

DISCRETIZATION

In the Multibody Dynamics interface you can choose not only the order of the discretization, but also the type of shape functions: *Lagrange* or *serendipity*. For highly distorted elements, Lagrange shape functions provide better accuracy than serendipity shape functions of the same order. The serendipity shape functions, however, give significant reductions of the model size for a given mesh containing hexahedral, prism, or quadrilateral elements.

The discretization order applies to the flexible bodies. The default is to use **Linear** shape functions for the **Displacement field**. If you want to compute stresses with good accuracy, increase the shape function order to **Quadratic serendipity** or **Quadratic**.



- [Domain, Boundary, Edge, Point, and Pair Nodes for the Multibody Dynamics Interface](#)
 - [Multibody Dynamics Theory](#)
-

Domain, Boundary, Edge, Point, and Pair Nodes for the Multibody Dynamics Interface

The **Multibody Dynamics Interface** has these domain, boundary, edge, point, and pair nodes (and subnodes), listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking then main physics interface node to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.



The links to the nodes described in external guides, such as the *Structural Mechanics Module User's Guide*, do not work in the PDF, only from the online help in COMSOL Multiphysics.

FEATURES AVAILABLE FROM SUBMENUS

Many features for the Multibody Dynamics interface are added from submenus in the Physics toolbar groups or context menu (when you right-click the node). The submenu name is the same in both cases.

The submenus at the **Domain** level are **Gears, Volume Forces, Mass, Spring, and Damper,** and **Domain Constraints**.

There are also the **Edges** and **Points** submenus.

The submenus at the **Global** level are **Joints**, and **Gear Pairs**.



Right-click the **Multibody Dynamics** node, select from the **Joints** menu (if **No Sorting** is selected from the context menu) or the **Global>Joints** menu (if **Sorting by Space Dimension** is selected).

The **Attachment** node is available from the **Physics** toolbar, **Boundaries** menu.

The following nodes are available from the **Physics** toolbar, **Domains** menu.

- Bevel Gear
- Helical Gear
- Helical Rack
- Initially Rigid
- Initial Values
- Rigid Domain
- Spur Gear
- Spur Rack
- Worm Gear

The following nodes are available from the **Physics** toolbar, **Global** menu.

- Ball Joint
- Base Motion
- Cam-Follower
- Cylindrical Joint
- Distance Joint
- Fixed Joint
- Gear Pair
- Hinge Joint
- Planar Joint
- Prismatic Joint
- Rack and Pinion
- Reduced Slot Joint
- Screw Joint
- Slot Joint
- Spring-Damper
- Universal Joint
- Worm and Wheel

These subnodes are available from the context menu (right-click the joint parent node) or from the **Physics** toolbar, **Attributes** menu:

- Applied Force and Moment
- Constraints
- Friction
- Locking
- Prescribed Motion
- Spring and Damper

These subnodes are automatically added to the joint parent node, or to one of its subnodes, when certain options are selected in the Settings window:

- Center of Joint Nodes
- Contact Area
- Destination Filter
- Destination Point Nodes
- Input Shaft Orientation
- Joint Axis
- Joint Elasticity
- Output Shaft Orientation
- Source Filter
- Source Point Nodes

These subnodes are automatically added to the Spring-Damper parent node when an appropriate option is selected in the Settings window:

- Destination Point (for Spring-Damper)
- Source Point (for Spring-Damper)

These subnodes are automatically added to the gear pair parent node when an appropriate option is selected in the Settings window:

- Backlash
- Friction (Gear)
- Gear Elasticity
- Transmission Error

These subnodes are automatically added to the gear or rack parent node when an appropriate option is selected in the Settings window:

- Gear Axis
- Rack Axis

The following nodes and subnodes are described for the Solid Mechanics interface in the *Structural Mechanics Module User's Guide* (listed in alphabetical order):

- Added Mass
- Applied Force (Rigid Connector)
- Applied Force (Rigid Domain)
- Applied Moment (Rigid Connector)
- Applied Moment (Rigid Domain)
- Body Load
- Boundary Load
- Change Thickness (2D)
- Center of Mass Nodes
- Center of Rotation Nodes
- Contact
- Damping
- Edge Load
- Fixed Constraint
- Fixed Constraint (Rigid Domain)
- Flexible Foundation
- Free
- Gravity
- Initial Stress and Strain
- Linear Elastic Material
- Location Nodes
- Mass and Moment of Inertia (Rigid Connector)
- Mass and Moment of Inertia (Rigid Domain)
- Moving Foundation
- Phase
- Point Load
- Predeformation
- Prescribed Displacement
- Prescribed Displacement/Rotation
- Radial Roller Bearing
- Rigid Connector
- Roller
- Rotating Frame
- Spring Foundation
- Thermal Expansion (for Materials)

Initial Values

The **Initial Values** node adds initial values for the displacement field and velocity field in a flexible component. The data can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear analysis. The initial values either can be directly prescribed in this node or can be inherited from the physics interface level. If needed, you can add more **Initial Values** nodes.

DOMAIN SELECTION

The domains, in which a **Rigid Domain** material model is defined, are automatically not applicable.

INITIAL VALUES

Select **From physics interface node** or **Locally defined**.

When using **From physics interface node**, the initial values for displacement field and velocity field are computed using the expressions entered for the rigid-body displacement, rotation, and velocities in the **Initial Values** section of the **Multibody Dynamics** node.

For **Locally defined**, enter values or expressions for the initial values of the **Displacement field \mathbf{u}** (the displacement components u , v , and w in 3D) and the **Structural velocity field $\partial u/\partial t$** .

Select a **Consistent initialization — Default** or **Force initial values**. The default when the given initial values are inconsistent is to make them adapt to each other by adjusting all initial values. To enforce that the exact values given in this node should be maintained through the consistent initialization process, use **Force initial values**. When **Force initial values** is selected, click to select the applicable check box or boxes — **Translation along first axis**, **Translation along second axis**, and **Translation along third axis**. Only the selected degrees of freedom have initial values fixed during the consistent initialization process.



- For details about the settings at the physics interface level, see [Initial Values](#).
- For details about the initialization process, see [Initialization in Time-Dependent Studies](#).

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Initial Values

Initially Rigid

Add the **Initially Rigid** node to enforce that a flexible domain is treated as a rigid body during the initialization of a time-dependent study.

This node has a domain selection and it can be only defined in domains where the **Linear Elastic Material** model is used. It overrides the **Initial Values** node and instead defines the initial values of the selected elastic domains itself.

INITIAL VALUES

Select **From physics interface node** or **Locally defined**.

- When **From physics interface node** is selected, initial values for rigid body displacement, rotation, and velocities are inherited from the physics interface level.
- When **Locally defined** is selected, an **Initial Values** subnode is automatically added, in which you can initialize the rigid domain degrees of freedom.

Select a **Consistent initialization** — **Default** or **Force initial values** to enforce that the exact values given in this node should be maintained through the consistent initialization process. The default when the given initial values are inconsistent is to make them adapt to each other by adjusting all initial values. For **Force initial values** click to select the applicable check boxes — **Translation along first axis**, **Translation along second axis**, **Translation along third axis**, and **Total rotation**. Only the selected degrees of freedom have the initial values fixed during the consistent initialization process.



- The optional subnode is the same as for a **Rigid Domain**, and is described in [Initial Values \(Rigid Domain\)](#).
- For details about the input expressions at the physics interface level, see [Initial Values](#).
- See [Rigid Domain Model](#) in the *Structural Mechanics Module User's Guide* for more information.
- For details about the initialization process, see [Initialization in Time-Dependent Studies](#).

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Initially Rigid

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Multibody Dynamics>Initially Rigid

Attachment

The **Attachment** node is used to define a set of boundaries on a flexible or rigid component, which can be used to connect it with other components through a joint or [Spring-Damper](#) in the Multibody Dynamics interface.

There are two formulations of the attachment. In the default formulation, all the selected boundaries behave as if they were connected by a common rigid body. In some cases, this give an unwanted stiffening or unrealistic local stresses. You can then switch to a flexible formulation where the constraint is applied only in an average sense.

Using attachments is optional when connecting a rigid component. It is, however, useful if you consider switching between a rigid and a flexible representation, since you then do not have to change the settings in the joints.

Attachments can be added to boundaries in a Multibody Dynamics or a Solid Mechanics interface, to edges in a Shell interface, and to points in a Beam interface. This makes it possible to use a joint for connecting to parts modeled in different physics interfaces.

CONNECTION TYPE

Select **Rigid** or **Flexible**.

When the connection type is rigid, the whole attachment selection acts as a virtual rigid object. This will cause a stiffening of the part with the attachment, and local stress concentrations at the selected boundaries if the domain with the attachment is flexible.

In the flexible formulation, the selected boundaries are allowed to have internal deformations, and the kinematic constraints are fulfilled only in an average sense. This will essentially provide a statically equivalent load distribution over the boundaries.

If a selected boundary is located on a rigid domain, the connection type setting does not matter. The rigid formulation is always used



With the flexible formulation, some extra degrees of freedom are added for each attachment. In order to get good convergence properties for a non-linear solution, these variables must have a good scaling.

When a new solver sequence is generated, then an appropriate manual scaling is automatically set for these variables.

If, however, an existing solver sequence was generated while **Connection Type** was set to **Rigid**, and you then change to **Flexible**, no such scaling will be present. In this case, you either have to regenerate the solver sequence, or set the scaling manually under the **Dependent Variables** node in each study step.



- [Attachments](#)
- [The Solid Mechanics Interface, The Shell and Plate Interfaces, and The Beam Interface](#) in the *Structural Mechanics Module User's Guide*.

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Attachment

Ribbon

Physics tab with **Multibody Dynamics** selected:

Boundaries>Multibody Dynamics>Attachment

Rigid Domain

Add the **Rigid Domain** node and select one or more domains to make them a rigid body.



These subnodes are described in the *Structural Mechanics Module User's Guide*:

- [Fixed Constraint \(Rigid Domain\)](#)
- [Prescribed Displacement/Rotation](#)
- [Applied Force \(Rigid Domain\)](#)
- [Applied Moment \(Rigid Domain\)](#)
- [Mass and Moment of Inertia \(Rigid Domain\)](#)
- [Spring Foundation \(Rigid Domain\)](#)

RIGID DOMAIN

The default **Density** ρ is taken **From material**. In this case, the material assignment for the domain supplies the mass density. For **User defined**, enter another value or expression.



The density is needed for dynamic analysis. It is also used when computing mass forces for gravitational or rotating frame loads, and when computing mass properties ([Computing Mass Properties](#)).

CENTER OF ROTATION

Select a **Center of Rotation** — **Center of mass**, **Centroid of selected entities**, or **User defined**. The center of rotation affects how displacements are interpreted and is also used as the default in various subnodes.

- For **Center of mass**, the center of rotation is taken as the center of mass of the rigid domain.
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge**, or **Point**. The available choices depend on geometrical dimension. The center of rotation is located at the centroid of the selected entities, which do not need to be related to rigid domain itself. As a special case, you can select a single point and thus use that point as center of rotation.



Once chosen, a default **Center of Rotation: Boundary**, **Center of Rotation: Edge**, or **Center of Rotation: Point** subnode is automatically added.

- For **User defined**, enter the **Global coordinates of center of rotation**, \mathbf{X}_c , in the table.

Select the **Offset** check box to add an optional offset vector to the definition of the center of rotation. Enter values for the offset vector $\mathbf{X}_{\text{offset}}$.

The center of rotation used is the sum of the vector obtained from any of the input methods and the offset vector.

$$\mathbf{X}_P = \mathbf{X}_{P,\text{input}} + \mathbf{X}_{\text{offset}}$$

INITIAL VALUES

Select **From physics interface node** or **Locally defined**.

- When **From physics interface node** is selected, initial values for rigid body displacement, rotation, and velocities are inherited from the physics interface level.
- When **Locally defined** is selected, an **Initial Values** subnode is automatically added, in which you can initialize the rigid domain degrees of freedom.

Select a **Consistent initialization** — **Default** or **Force initial values** to enforce that the exact values given in this node should be maintained through the consistent initialization process. The default when the given initial values are inconsistent is to make them adapt to each other by adjusting all initial values. For **Force initial values**, click to select the applicable check boxes — **Translation along first axis**, **Translation along second axis**,

Translation along third axis, and **Total rotation**. Only the selected degrees of freedom have the initial values fixed during the consistent initialization process.



- The optional subnode is described in [Initial Values \(Rigid Domain\)](#).
- For details about the input expressions at the physics interface level, see [Initial Values](#).
- See [Rigid Domain Model](#) in the *Structural Mechanics Module User's Guide* for more information.
- For details about the initialization process, see [Initialization in Time-Dependent Studies](#).

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Rigid Domain

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Multibody Dynamics>Rigid Domain

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Multibody Dynamics>Initial Values

Center of Rotation Nodes

For each of the following physics features, a subnode is automatically added as indicated.

- [Rigid Domain](#)
- [Initial Values \(Rigid Domain\)](#)
- Gears, as described under [Spur Gear](#)

CENTER OF ROTATION: BOUNDARY

Use the **Center of Rotation: Boundary** subnode to select a set of boundaries whose centroid represents the center of rotation.

CENTER OF ROTATION: EDGE

Use the **Center of Rotation: Edge** subnode to select a set of edges whose centroid represents the center of rotation.

CENTER OF ROTATION: POINT

Use the **Center of Rotation: Point** subnode to select a set of points whose centroid represents the center of rotation.

LOCATION IN USER INTERFACE

These nodes are automatically added when **Centroid of selected entities** is selected in the parent node. They cannot be added or removed manually.

Initial Values (Rigid Domain)

The **Initial Values** subnode is automatically added when you select that the initial values in a **Rigid Domain** node are **Locally defined**.

You can enter initial values for the rigid body displacement, rotation and velocities, which can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear analysis. The initial values that you specify are interpreted in the selected coordinate system.

CENTER OF ROTATION

Select an option from the list: **From parent**, **Centroid of selected entities**, or **User defined**. The given initial values are interpreted at the center of rotation.

- For **From parent**, the center of rotation is taken as the one defined in the parent **Rigid Domain** node.
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge**, or **Point**. The available choices depend on geometrical dimension. The initial conditions act at the centroid of the selected entities, which do not need to be related to rigid domain itself. As a special case, you can select a single point and thus prescribe the initial conditions at that point.



Once chosen, a default **Center of Rotation: Boundary**, **Center of Rotation: Boundary**, or **Center of Rotation: Point** subnode is automatically added.

- For **User defined**, enter the **Global coordinates of center of rotation**, \mathbf{X}_c , in the table.

Select the **Offset** check box to add an optional offset vector to the definition of the center of rotation. Enter values for the offset vector $\mathbf{X}_{\text{offset}}$.

The center of rotation used is the sum of the vector obtained from any of the input methods and the offset vector.

$$\mathbf{X}_P = \mathbf{X}_{P,\text{input}} + \mathbf{X}_{\text{offset}}$$

INITIAL VALUES: TRANSLATIONAL

- **Displacement at center of rotation** \mathbf{u} .
- **Velocity at center of rotation** $\partial\mathbf{u}/\partial t$.

INITIAL VALUES: ROTATIONAL

- *For 3D components only:* **Axis of rotation** Ω
- **Angle of rotation** ϕ .
- **Angular velocity** ω (3D components) and $\partial\phi/\partial t$ (2D components)

LOCATION IN USER INTERFACE

This node is automatically added when **Locally defined** is selected in the **Initial Values** section in the parent **Rigid Domain** node. It cannot be added or removed manually.

Prismatic Joint

Use the **Prismatic Joint** node for 2D and 3D components to create a joint between two components such that they are free to translate relative to each other along the joint axis.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint.

ATTACHMENT SELECTION

Select a **Source** and a **Destination** attachment from the lists. You can select any [Attachment](#), [Rigid Domain](#), [Spur Gear](#), [Helical Gear](#), [Bevel Gear](#), [Worm Gear](#), [Spur Rack](#), or [Helical Rack](#) that is defined for the model. In **Source** an additional option, **Fixed**, is also available. When **Fixed** is selected, the source side of the joint is fixed in space, and not connected to any modeled parts. If any [Base Motion](#) nodes are present in the model, these also appear in the list for the **Source** attachment. Selecting a base motion gives a behavior similar to using **Fixed**, with the exception that the base now can have a nonzero prescribed displacement, velocity, or acceleration.

In large models, the list of available attachments can become long. You can then create geometric filters to narrow down the search. When the **Use selection filter** check box is selected, two subnodes named **Source Filter** and **Destination Filter** are added to the joint node. In these subnodes you can make graphic selections of the objects whose attachments should be shown in the **Source** and a **Destination** lists.

CENTER OF JOINT

The center of joint is the point about which displacements and rotations of source and destination boundaries are interpreted. Select an option from the list — **Centroid of source**, **Centroid of destination**, **Centroid of selected entities**, or **User defined**.

- For **Centroid of source** and **Centroid of destination**, the joint is located at the centroid of the selection giving the source or destination, respectively.
- For **Center of mass**, the center of joint is taken as the center of mass of the gear.
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge**, or **Point**. The available choices depend on geometrical dimension. The center of joint is located at the centroid of the selected entities. As a special case, you can select a single point, and thus use that point as center of joint.



Once chosen, a default **Center of Joint: Boundary**, **Center of Joint: Edge**, or **Center of Joint: Point** subnode is automatically added.

- For **User defined**, enter the **Global coordinates of center of rotation**, \mathbf{X}_c , in the table.

Select the **Offset** check box to add an optional offset vector to the definition of the center of rotation. Enter values for the offset vector $\mathbf{X}_{\text{offset}}$.

The center of rotation used is the sum of the vector obtained from any of the input methods and the offset vector.

$$\mathbf{X}_P = \mathbf{X}_{P,\text{input}} + \mathbf{X}_{\text{offset}}$$

AXIS OF JOINT

The axis of joint is the axis along which the relative motion is allowed. Select an option from the list — **Specify direction**, **Select a parallel edge**, or **From selected coordinate system**.

- For **Specify direction** enter a value or expression for \mathbf{e}_0 . The default is (1, 0, 0). The direction is specified in the selected coordinate system.

- For **Select a parallel edge** an edge can be selected on the [Joint Axis](#) subnode, which is added automatically. The vector from the first to the last end of the edge is used to define the joint axis. Any edge in the model can be used. Select the **Reverse direction** check box to reverse the direction of the joint axis.
- For **From selected coordinate system** the joint axis is formed using the selected coordinate system. With **Axis to use** you select the axis (first, second, or third) that is parallel to the joint axis.

JOINT ELASTICITY

Select an option from the list to determine whether the joint should be fully rigid or elastic — **Rigid joint** or **Elastic joint**. For **Elastic joint**, the properties of an elastic joint are given in the [Joint Elasticity](#) subnode, which is added automatically.

JOINT FORCES AND MOMENTS

Select an option from the list to use the method of computation for the forces and moments — **Computed on destination**, **Computed on source**, **Do not compute**, **Computed using weak constraints**, or **Computed using penalty method**. For **Computed using penalty method**, enter the **Penalty factor** p_j . The default is $(mbd.prj1.Eequ*(0.1*mbd.diag))$. The exact naming of the variable depends on the tags of the nodes in the model tree.



- [Prismatic Joint Theory](#)
 - [Computing Joint Forces and Moments](#)
-

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Prismatic Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Prismatic Joint

Hinge Joint

Use the **Hinge Joint** node for 2D and 3D components to create a joint between two components such that they are free to rotate relative to each other about the joint axis at the center of joint.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint.



See [Prismatic Joint](#) for all settings — **Attachment Selection**, **Center of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**. For 3D components, also see **Axis of Joint**.



[Hinge Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Hinge Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Hinge Joint

Cylindrical Joint

Use the **Cylindrical Joint** node for 3D components to create a joint between two components such that they are free to translate relative to each other along the joint axis and also free to rotate relative to each other about the same axis.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint



See [Prismatic Joint](#) for all settings — **Attachment Selection**, **Center of Joint**, **Axis of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**.



[Cylindrical Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Cylindrical Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Cylindrical Joint

Screw Joint

Use the **Screw Joint** node for 3D components to create a joint between two components such that they are free to rotate relative to each other about the joint axis and at the same time translate along the same axis. The translation and rotation are not independent; they are connected through the number of starts and the pitch of a screw thread.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint.

THREAD PARAMETERS

Enter a value or expression for the **Pitch** p and **Start** n .



See [Prismatic Joint](#) for these settings — **Attachment Selection**, **Center of Joint**, **Axis of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**.



[Screw Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Screw Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Screw Joint

Planar Joint

Use the **Planar Joint** node for 3D components to create a joint between two components such that they are free to translate relative to each other in a plane perpendicular to the joint axis and also free to rotate relative to each other about the same axis.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint.



See [Prismatic Joint](#) for all settings — **Attachment Selection**, **Center of Joint**, **Axis of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**.



[Planar Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Planar Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Planar Joint

Ball Joint

Use the **Ball Joint** node for 3D components to create a joint between two components such that they are free to rotate relative to each other about all three axes at the center of joint.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), [Applied Force and Moment](#), and [Friction](#) subnodes can be added to modify the properties of the joint.

AXES OF JOINT

Enter values or expressions for the **Source axis pointing toward center of joint** \mathbf{e}_0 and **Destination axis pointing away from center of joint** \mathbf{e}_{d0} . The default is (1, 0, 0) for both axes.



See [Prismatic Joint](#) for these settings — **Attachment Selection, Center of Joint, Joint Elasticity, Joint Forces and Moments, and Sketch.**



[Ball Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Ball Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Ball Joint

Slot Joint

Use the **Slot Joint** node for 3D components to create a joint between two components such that they are free to rotate relative to each other about all three axes at the center of joint and also free to translate relative to each other along the joint translational axis.

The [Constraints](#), [Locking, Spring and Damper](#), [Prescribed Motion](#), and [Applied Force and Moment](#) subnodes can be added to modify the properties of the joint.

AXES OF JOINT

Enter values or expressions for the **Source axis pointing toward center of joint** \mathbf{e}_0 and **Destination axis pointing away from center of joint** \mathbf{e}_{d0} . The default is (1, 0, 0) for both axes. Select an option from the **Joint translation axis** list — **Destination axis** or **Source axis**.



See [Prismatic Joint](#) for these settings — **Attachment Selection, Center of Joint, Joint Elasticity, Joint Forces and Moments, and Sketch.**



LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Slot Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Slot Joint

Reduced Slot Joint

Use the **Reduced Slot Joint** node for 2D and 3D components to create joints between two components such that they are free to rotate relative to each other about the joint rotational axis and also free to translate relative to each other along the joint translational axis.

The [Constraints](#), [Locking](#), [Spring and Damper](#), [Prescribed Motion](#), and [Applied Force and Moment](#) subnodes can be added to modify the properties of the joint.

AXES OF JOINT



For 3D components, Select the **Joint translational axis — Attached on destination** or **Attached on source** then enter values or expressions for **Joint translation axis** \mathbf{e}_{tr0} and **Joint rotational axis** \mathbf{e}_{rot0} . The default is (1,0,0) and (0,1,0) for \mathbf{e}_{tr0} and \mathbf{e}_{rot0} , respectively.



For 2D components, select the **Joint translational axis — Attached on destination** or **Attached on source** then enter values or expressions for **Joint translation axis** \mathbf{e}_{tr0} . The default is (1,0,0).



See [Prismatic Joint](#) for these settings — **Attachment Selection**, **Center of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**.



Reduced Slot Joint Theory

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>Reduced Slot Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Joints>Reduced Slot Joint

Fixed Joint

Use a **Fixed Joint** for 2D and 3D components to prevent any relative motion between the two components. The components connected by this joint are rigidly attached to each other and no additional degrees of freedom are added. The forces or moments required to restrict the relative motion between the two components can be computed.



See [Prismatic Joint](#) for all settings — **Attachment Selection, Joint Elasticity, Joint Forces and Moments, Center of Joint,** and **Axis of Joint.**



Fixed Joint Theory

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>More Joints>Fixed Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>More Joints>Fixed Joint

Distance Joint

Use a **Distance Joint** for 2D and 3D components to maintain a specified distance between two components. The two components behave as if they are connected through a rigid massless link with ball joints at the ends. No additional degrees of freedom are added by this joint.

It is also possible to specify a change in distance by prescribing the extension. This can be interpreted as if, instead of a rigid link, there is telescopic link connecting the two components.

ATTACHMENT SELECTION

Select a **Source**. Select any [Attachment](#), [Rigid Domain](#), [Spur Gear](#), [Helical Gear](#), [Bevel Gear](#), [Worm Gear](#), [Spur Rack](#), [Helical Rack](#), or [Base Motion](#) that is defined for the model. The option **Fixed** is also available. When **Fixed** is selected, the source side of the joint is fixed in space, and not connected to any modeled parts. Selecting a base motion gives a behavior similar to using **Fixed**, with the exception that the base now can have a nonzero prescribed displacement, velocity or acceleration.

Select a **Connection point** — **Centroid of source**, **Centroid of selected entities**, or **User defined**.

- For **User defined** also enter the global coordinates of the connection point \mathbf{X}_g .
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge** (3D components), or **Point**.



Once chosen, a default [Source Point: Boundary](#), [Source Point: Edge](#), or [Source Point: Point](#) subnode is added.

The connection point is the exact location where the fictitious ball joint is connected to the source attachment. If the source is **Fixed**, the only option available is **User defined**.

Select a **Destination**. Select any [Attachment](#), [Rigid Domain](#), [Spur Gear](#), [Helical Gear](#), [Bevel Gear](#), [Worm Gear](#), [Spur Rack](#), or [Helical Rack](#) that is defined for the model.

Select a **Connection point** — **Centroid of destination**, **Centroid of selected entities**, or **User defined**.

- For **User defined** also enter the global coordinates of the connection point \mathbf{X}_d .
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge** (3D components), or **Point**.



Once chosen, a default [Destination Point: Boundary](#), [Destination Point: Edge](#), or [Destination Point: Point](#) subnode is added.

In large models, the list of available attachments can become long. You can then create geometric filters to narrow down the search. When the **Use selection filter** check box is selected, two subnodes named [Source Filter](#) and [Destination Filter](#) are added to the joint node. In these subnodes you can make graphic selections of the objects whose attachments should be shown in the **Source** and a **Destination** lists.

MODIFY DISTANCE

Enter an **Extension** Δ . The default is 0, in which case the distance is equal to the initial geometrical distance between the two connection points.



[Distance Joint Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>More Joints>Distance Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>More Joints>Distance Joint

Universal Joint

A universal joint, also known as Hooke's joint or Cardan joint, is commonly used to transmit rotational motion between shafts that have different orientations. You can use the **Universal Joint** node in 3D components to simulate this behavior. Physically, this joint is made up of pair of hinge joints, oriented perpendicular to each other,

connected by a cross shaft or gimbal. The universal joint is an abstract joint where the gimbal is not modeled and both the shafts are directly connected to each other. In general, this joint is used for the purpose of transfer of mechanical power from one shaft to another shaft. The added degrees of freedom in this joint are the relative rotations about the two legs of the gimbal that form joint axis and second axis.



See [Prismatic Joint](#) for these settings — **Attachment selection**, **Center of Joint**, **Joint Elasticity**, **Joint Forces and Moments**, and **Sketch**.

SHAFTS ORIENTATION

Select an **Input shaft orientation** — **Specify direction** or **Select a parallel edge**.

- For **Specify direction** enter a value or expression for \mathbf{e}_{i0} . The default is (1, 0, 0). The direction is given in the coordinate system selected in the **Coordinate system selection** section.
- For **Select a parallel edge** an edge can be selected on the [Input Shaft Orientation](#) subnode, which is added automatically. The vector from the first to the last end of the edge is used to define the input shaft orientation. Any edge in the model can be used. Select the **Reverse direction** check box to reverse the direction of the shaft orientation.

Select an **Output shaft orientation** — **Specify direction** or **Select a parallel edge**.

- For **Specify direction** enter a value or expression for \mathbf{e}_{o0} . The default is (1, 1, 0). The direction is given in the coordinate system selected in the **Coordinate system selection** section.
- For **Select a parallel edge** an edge can be selected on the [Output Shaft Orientation](#) subnode, which is added automatically. The vector from the first to the last end of the edge is used to define the output shaft orientation. Any edge in the model can be used. Select the **Reverse direction** check box to reverse the direction of the shaft orientation.

GIMBAL ORIENTATION

Select an **Auxiliary axis** — **Perpendicular to the plane of shafts** or **User defined**. For **User defined** enter a value or expression for \mathbf{e}_{aux0} based on space dimension. The direction is given in the coordinate system selected in the **Coordinate system selection** section.

Select an **Axis connecting Gimbal and Input shaft** — **Same as auxiliary axis** or **Rotated auxiliary axis**. For **Rotated auxiliary axis** enter a value or expression for the **Rotation about input shaft** θ_{g0} .



Universal Joint Theory

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Joints>More Joints>Universal Joint

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>More Joints>Universal Joint

Joint Elasticity

The **Joint Elasticity** subnode is automatically added to a joint node when **Elastic joint** is chosen under **Joint Elasticity**. You use it to specify elastic and viscous data for the constrained degrees of freedom in a joint.

The contents of the **Joint Elasticity** settings adapt to the type of joint used.



For specifying elastic or viscous properties for the degrees of freedom which are not constrained by the joint, see [Spring-Damper](#).

COORDINATE SYSTEM SELECTION

Select a coordinate system in which the elasticity and viscosity matrices are described. The default is **Joint coordinate system**, in which case the directions are the same as those specified for the joint itself.

ELASTIC DEGREES OF FREEDOM

Select the degrees of freedom that are to be treated as elastic. The default is all constrained degrees of freedom for this type of joint. If you deselect a certain degree of freedom, it reverts to being treated as rigidly constrained. The axis directions always refer to the directions of the joint coordinate system, irrespective of the settings in the **Coordinate System Selection** section.

Check boxes are shown only for the degrees of freedom which are constrained by the current joint type.

SPRING

Specify the elastic properties of the joint. For joints constraining both translational and rotational degrees of freedom, select **Translational-rotational coupling** to enable input of stiffness terms coupling translation and rotation.

- In \mathbf{k}_u , enter the translational stiffness matrix.
- In \mathbf{k}_θ , enter the rotational stiffness matrix.
- For **Translational-rotational coupling** also enter the coupling matrices $\mathbf{k}_{u\theta}$ and $\mathbf{k}_{\theta u}$.

VISCOUS DAMPING

In this section, you specify the viscous damping properties of the joint. For joints constraining both translational and rotational degrees of freedom, select **Translational-rotational coupling** to enable input of viscous terms coupling translation and rotation.

- In \mathbf{c}_u , enter the translational damping matrix.
- In \mathbf{c}_θ , enter the rotational damping matrix.
- For **Translational-rotational coupling** also enter the coupling matrices $\mathbf{c}_{u\theta}$ and $\mathbf{c}_{\theta u}$.



The stiffness and damping matrices are always entered with the largest possible size (for example as 3x3 matrices in 3D). Values entered in directions that are not part of the degrees of freedom declared as elastic are ignored.



The rotational and coupling matrix inputs are not available for the **Ball Joint**, **Slot Joint**, **Hinge Joint** in 2D, and **Reduced Slot Joint** in 2D. These joint do not have any rotational constraints.



[Elasticity in Joints](#)

LOCATION IN USER INTERFACE

This node is automatically added when **Elastic joint** is selected in the parent joint node. It cannot be added or removed manually.

Constraints

The **Constraints** subnode is used to impose a restriction on the relative motion between two components. You can specify an upper and lower bound of relative motion. This node can be added more than once if required, and the most restrictive value will then take precedence.

The **Constraints** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#). The contents of the settings sections will differ between different joint types. Multiple sections can be present in a single joint, depending on the number of degrees of freedom it has.



The [Theory for Joint Constraints](#) section describes the theory for each of the different types of constraints: translational, rotational, inclination angle, and axial rotation.

TRANSLATIONAL CONSTRAINTS

This section is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) nodes.

- For the **Planar Joint** only, enter values or expressions for the **Initial constraint axis** e_{c0} .
- Enter the **Maximum relative displacement** u_{\max} .
- Enter the **Minimum relative displacement** u_{\min} .
- Select **Constraint parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** p_u . The default is $((1[1/ms])^2) * (mbd.prj1.ct1.me)$. The exact naming of the variable depends on the tags of the nodes in the model tree.
- Enter a value or expression for the **Maximum allowable time step** Δt_{\max} .

The default is

$((\pi) / (30)) * (\text{sqrt}((mbd.prj1.ct1.me) / (mbd.prj1.ct1.p_u)))$. The exact naming of the variables depends on the tags of the nodes in the model tree.

ROTATIONAL CONSTRAINTS

This section is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#).

- Enter the **Maximum relative rotation** θ_{\max} .
- Enter the **Minimum relative rotation** θ_{\min} .
- Select **Constraint parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** p_{θ} . The default is $((1[1/\text{ms}])^2) * (\text{mbd.hg}j1.\text{ct}1.\text{Ie})$. The exact naming of the variable depends on the tags of the nodes in the model tree.
- Enter a value or expression for the **Maximum allowable time step** Δt_{\max} .
The default is $((\pi)/(30)) * (\text{sqrt}((\text{mbd.hg}j1.\text{ct}1.\text{Ie})/(\text{mbd.hg}j1.\text{ct}1.\text{p_th})))$. The exact naming of the variables depends on the tags of the nodes in the model tree.

INCLINATION ANGLE CONSTRAINT

This section is available for the [Ball Joint](#) and [Slot Joint](#).

- Select the **Initial reference axis** — **Initial source axis**, **Initial destination axis**, or **User defined**. For **User defined** enter values for \mathbf{e}_{R0} .
- Enter a value or expression for the **Maximum inclination angle** θ_{\max} .
- Select the **Constraint parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** p_{θ} . The default is $((1[1/\text{ms}])^2) * (\text{mbd.bl}j1.\text{ct}1.\text{Ie_th})$. The exact naming of the variable depends on the tags of the nodes in the model tree.
- Enter a value or expression for the **Maximum allowable time step** Δt_{\max} .
The default is $((\pi)/(30)) * (\text{sqrt}((\text{mbd.bl}j1.\text{ct}1.\text{Ie_th})/(\text{mbd.bl}j1.\text{ct}1.\text{p_th})))$. The exact naming of the variables depends on the tags of the nodes in the model tree.

AXIAL ROTATION CONSTRAINTS

This section is available for the [Ball Joint](#) and [Slot Joint](#).

- Enter the **Maximum axial rotation** ψ_{\max} .
- Enter the **Minimum axial rotation** ψ_{\min} .
- Select the **Constraint parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** p_{ψ} . The default is

$((1[1/ms])^2) * (mbd.blj1.ct1.Ie_psi)$. The exact naming of the variable depends on the tags of the nodes in the model tree.

- Enter a value or expression for the **Maximum allowable time step** Δt_{max} .

The default is

$((\pi) / (30)) * (\text{sqrt}((mbd.blj1.ct1.Ie_psi) / (mbd.blj1.ct1.p_psi)))$.

The exact naming of the variables depends on the tags of the nodes in the model tree.

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Prismatic Joint>Constraints

Multibody Dynamics>Hinge Joint>Constraints

Multibody Dynamics>Cylindrical Joint>Constraints

Multibody Dynamics>Screw Joint>Constraints

Multibody Dynamics>Planar Joint>Constraints

Multibody Dynamics>Ball Joint>Constraints

Multibody Dynamics>Slot Joint>Constraints

Multibody Dynamics>Reduced Slot Joint>Constraints

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Ball Joint**, **Slot Joint**, or **Reduced Slot Joint** selected in the model tree:

Attributes>Constraints

Locking

Use the **Locking** subnode to put a lock condition on the relative motion between two components on a joint. Lock conditions are similar to [Constraints](#), but once the displacement in the joint degree of freedom reaches the limiting value, it will be fixed to that value. You can specify the locking value for the relative motion in the form of upper and lower bound. This node can be added more than once if required, and the most restrictive value will then take precedence.

The **Locking** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#).

If multiple degrees of freedom are present in a joint, a lock condition can be put on each of the degrees of freedom. If all the degrees of freedom are locked, the

components attached to the joint move as a single object. The contents of the settings sections will differ between different joint types.



The [Theory for Joint Locking](#) section describes the theory for each of the different types of locks: translational, rotational, inclination angle, and axial rotation.

TRANSLATIONAL LOCKING

This section is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) nodes.

- For the **Planar Joint** only, enter values or expressions for the **Initial locking axis** \mathbf{e}_{10} .
- Enter the **Maximum relative displacement** u_{\max} .
- Enter the **Minimum relative displacement** u_{\min} .
- Select the **Locking parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** p_{u1} , which supplies a penalty stiffness used for enforcing the constraint. The default is $((0.01) * (\text{mbd.prj1.lk1.Eequ})) * (\text{mbd.diag})$. Enter a value or expression for the **Penalty factor** p_{u2} , which supplies damping suppressing vibrations. The default is $(\text{mbd.prj1.lk1.p_u1}) * (10[\text{ms}])$. The exact naming of the variables depends on the tags of the nodes in the model tree.

ROTATIONAL LOCKING

This section is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#).

- Enter the **Maximum relative rotation** θ_{\max} .
- Enter the **Minimum relative rotation** θ_{\min} .
- Select the **Locking parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** $p_{\theta1}$.
The default is $((1e-4) * (\text{mbd.hgj1.lk1.Eequ})) * ((\text{mbd.diag})^{\wedge}(3))$. The exact naming of the variables depends on the tags of the nodes in the model tree. Enter a value or expression for the **Penalty factor** $p_{\theta2}$. The default is $(\text{mbd.hgj1.lk1.p_th1}) * (10[\text{ms}])$. The exact naming of the variable depends on the tags of the nodes in the model tree.

INCLINATION ANGLE LOCKING

This section is available for the [Ball Joint](#) and [Slot Joint](#).

- Select the **Initial reference axis** — **Initial source axis**, **Initial destination axis**, or **User defined**. For **User defined** enter values for \mathbf{e}_{R0} .
- Enter a value or expression for the **Maximum inclination angle** θ_{\max} .
- Select the **Locking parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** $p_{\theta 1}$.

The default is $((1e-4) * (\text{mbd.blj1.lk1.Eequ})) * ((\text{mbd.diag})^{\wedge}(3))$. The exact naming of the variables depends on the tags of the nodes in the model tree. Enter a value or expression for the **Penalty factor** $p_{\theta 2}$. The default is $(\text{mbd.blj1.lk1.p_th1}) * (10[\text{ms}])$. The exact naming of the variable depends on the tags of the nodes in the model tree.

AXIAL ROTATION LOCKING

This section is available for the [Ball Joint](#) and [Slot Joint](#).

- Enter the **Maximum axial rotation** ψ_{\max} .
- Enter the **Minimum axial rotation** ψ_{\min} .
- Select the **Locking parameters** — **Automatic** or **User defined**. For **User defined** enter a value or expression for the **Penalty factor** $p_{\psi 1}$.

The default is $(1e-4) * (\text{mbd.blj1.lk1.Eequ}) * (\text{mbd.diag})^{\wedge}(3)$. The exact naming of the variables depends on the tags of the nodes in the model tree. Enter a value or expression for the **Penalty factor** $p_{\psi 2}$. The default is $(\text{mbd.blj1.lk1.p_psi1}) * (10[\text{ms}])$. The exact naming of the variable depends on the tags of the nodes in the model tree.

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Prismatic Joint>Locking

Multibody Dynamics>Hinge Joint>Locking

Multibody Dynamics>Cylindrical Joint>Locking

Multibody Dynamics>Screw Joint>Locking

Multibody Dynamics>Planar Joint>Locking

Multibody Dynamics>Ball Joint>Locking

Multibody Dynamics>Slot Joint>Locking

Multibody Dynamics>Reduced Slot Joint>Locking

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Ball Joint**, **Slot Joint**, or **Reduced Slot Joint** selected in the model tree:

Attributes>Locking

Spring and Damper

Use the **Spring and Damper** subnode to apply elastic and viscous forces to restrict and control the relative motion on joint. You can also use a dashpot to model various kind of losses in the joint. This node can be added more than once if required, and the effects will then be superimposed. Each joint type has different settings sections.

The **Spring and Damper** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#). The contents of the settings sections will differ between different joint types



The [Theory for Springs and Dampers on Joints](#) section describes the theory for each of the different types of springs and dampers: translational, rotational, inclination angle, and axial rotation.

SPRING AND DAMPER: TRANSLATIONAL

This section is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) nodes.

Enter values or expressions for the following:

- **Spring constant** k_u .
- **Predeformation** u_0 .
- **Damping coefficient** c_u .

For the **Planar Joint**, enter values or expressions for the **Initial spring axis** \mathbf{e}_{s0} . Also select the **Initial damper axis** — **Same as spring axis** or **User defined**. For **User defined** enter values or expressions for \mathbf{e}_{d0} .

SPRING AND DAMPER: ROTATIONAL

This section is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Reduced Slot Joint](#) nodes.

Enter values or expressions for the following:

- **Spring constant** k_θ .

- **Predeformation** θ_0 .
- **Damping coefficient** c_θ .

SPRING AND DAMPER: INCLINATION ANGLE

This section is available for [Ball Joint](#) and [Slot Joint](#) nodes.

Select the **Initial reference axis** — **Initial source axis**, **Initial destination axis**, or **User defined**. For **User defined** enter values for \mathbf{e}_{R0} .

Enter values or expressions for the following:

- **Spring constant** k_θ .
- **Predeformation** θ_0 .
- **Damping coefficient** c_θ .

SPRING AND DAMPER: AXIAL ROTATION

This section is available for [Ball Joint](#) and [Slot Joint](#) nodes.

Enter values or expressions for the following:

- **Spring constant** k_ψ .
- **Predeformation** ψ_0 .
- **Damping coefficient** c_ψ .

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Prismatic Joint>Spring and Damper
Multibody Dynamics>Hinge Joint>Spring and Damper
Multibody Dynamics>Cylindrical Joint>Spring and Damper
Multibody Dynamics>Screw Joint>Spring and Damper
Multibody Dynamics>Planar Joint>Spring and Damper
Multibody Dynamics>Ball Joint>Spring and Damper
Multibody Dynamics>Slot Joint>Spring and Damper
Multibody Dynamics>Reduced Slot Joint>Spring and Damper

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Ball Joint**, **Slot Joint**, or **Reduced Slot Joint** selected in the model tree:
Attributes>Spring and Damper

Prescribed Motion

Use the **Prescribed Motion** subnode to prescribe the relative motion on a joint. You can also prescribe the motion by giving the time derivatives of the joint degrees of freedom, for example, velocity for translation motion and angular velocity for rotational motion. Activation conditions can be used to prescribe the motion for only a certain duration. Axis directions are interpreted in the selected coordinate system.

The **Prescribed Motion** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#). The contents of the settings sections will differ between different joint types



The [Theory for Prescribed Motion on Joints](#) section describes the theory for each of the different types of prescribed motion: translational and rotational.

PRESCRIBED TRANSLATIONAL MOTION

This section is available for the [Prismatic Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) nodes.

Select an option from the **Prescribed motion through** list — **Displacement** or **Velocity**.

- For **Displacement** enter a value or expression for u_p .
- For **Velocity** enter a value or expression for v_p .

Select an **Activation condition** — **Always active** or **Conditionally active**. For **Conditionally active** enter a **Deactivation indicator expression** i_{up} . For the joints having more than one degree of freedom ([Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#)), **Never active** is also an option.

For the **Planar Joint**, enter values or expressions for the **Initial prescribed motion axis** e_{p0} .

PRESCRIBED ROTATIONAL MOTION

This section is available for the [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#) nodes.

For the **Ball Joint** and **Slot Joint**, enter values or expressions for the **Initial prescribed motion axis** e_{p0} .

Select an option from the **Prescribed motion through** list — **Rotation** or **Angular velocity**.

- For **Rotation** enter a value or expression for θ_p . In this case, the variable name is ϕ_p for the Ball Joint and Slot Joint.
- For **Angular velocity** enter a value or expression for ω_p .

Select an **Activation condition** — **Always active** or **Conditionally active**. For **Conditionally active** enter a **Deactivation indicator expression** $i_{\theta p}$. (In this case, the variable name is $i_{\phi p}$ for the Ball Joint and Slot Joint). For the joints having more than one degree of freedom (Cylindrical Joint, Screw Joint, Planar Joint, Slot Joint, and Reduced Slot Joint), **Never active** is also an option.

REACTION FORCE SETTINGS

Select **Evaluate reaction forces** to compute the reaction force caused by the prescribed motion. The default is to not compute the reaction force. When selected, the prescribed motion is implemented as a weak constraint. If the prescribed motion is not active all through the study step, because **Activation condition** is **Conditionally active**, this setting is ignored. Reaction forces are never computed in that case.

Select **Apply reaction only on joint variables** to use a unidirectional constraint for enforcing the prescribed motion. The default is that bidirectional constraints are used. This setting is useful in a situation where a bidirectional constraint would give an unwanted coupling in the equations. This would happen if the prescribed value of the motion is a variable solved for in other equations.

If the motion is prescribed as a velocity, the reaction force is applied on the joint variables only (unidirectional constraint) irrespective of the setting of this control.

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Prismatic Joint>Prescribed Motion

Multibody Dynamics>Hinge Joint>Prescribed Motion

Multibody Dynamics>Cylindrical Joint>Prescribed Motion

Multibody Dynamics>Screw Joint>Prescribed Motion

Multibody Dynamics>Planar Joint>Prescribed Motion

Multibody Dynamics>Ball Joint>Prescribed Motion

Multibody Dynamics>Slot Joint>Prescribed Motion

Multibody Dynamics>Reduced Slot Joint>Prescribed Motion

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Ball Joint**, **Slot Joint**, or **Reduced Slot Joint** selected in the model tree:

Attributes>Prescribed Motion

Applied Force and Moment

Use the **Applied Force and Moment** subnode to apply forces and moments on either the source or destination attachment of a joint, or directly on the joint degrees of freedom.

The **Applied Force and Moment** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), [Ball Joint](#), [Slot Joint](#), and [Reduced Slot Joint](#). The contents of the settings sections will differ between different joint types.

APPLIED ON

Select **Source attachment**, **Destination attachment**, or **Joint** to apply force and moment on the selected attachments or directly on the joint degrees of freedom.

LOCATION

In this section you define the point of action of the force. This section is not available when the force is applied on **Joint**.

Select **Center of attachment**, **Center of joint**, or **User defined**. For **User defined** enter values or expressions for the **Location** \mathbf{X}_p .

APPLIED FORCE AND MOMENT

For **Source attachment** or **Destination attachment**, enter values or expressions for the applied force \mathbf{F} and applied moment \mathbf{M} (3D components) or M_z . The forces or moments that you prescribe are interpreted in the selected coordinate system.

When the load is applied on **Joint**, the input in this section depends on the joint type. For all joints, a scalar force or moment acts directly on the single translational or rotational degree of freedom in the joint, while vector inputs have directions as given by the selected coordinate system.

- For **Prismatic Joint**, enter the scalar force F .
- For **Hinge Joint**, enter the scalar moment M .
- For **Cylindrical Joint**, **Screw Joint** or **Reduced Slot Joint**, enter the scalar force F and moment M .

- For **Planar Joint**, enter the force vector \mathbf{F} , and the scalar moment M .
- For **Ball Joint**, enter the moment vector \mathbf{M} .
- For **Slot Joint**, enter the scalar force F and the moment vector \mathbf{M} .



Theory for Applied Forces and Moments on Joints

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Prismatic Joint>Applied Force and Moment

Multibody Dynamics>Hinge Joint>Applied Force and Moment

Multibody Dynamics>Cylindrical Joint>Applied Force and Moment

Multibody Dynamics>Screw Joint>Applied Force and Moment

Multibody Dynamics>Planar Joint>Applied Force and Moment

Multibody Dynamics>Ball Joint>Applied Force and Moment

Multibody Dynamics>Slot Joint>Applied Force and Moment

Multibody Dynamics>Reduced Slot Joint>Applied Force and Moment

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, **Ball Joint**, **Slot Joint**, or **Reduced Slot Joint** selected in the model tree:

Attributes>Applied Force and Moment

Friction

Use the **Friction** subnode to model friction losses in the joint.

The **Friction** subnode can be added to the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), [Planar Joint](#), and [Ball Joint](#).



Friction can only be applied to the joints in a time-dependent analysis.

FRICITION

Enter a **Frictional coefficient** μ .

Enter a **Characteristic slip velocity** v_0 . The default is `mbd.diag*(1e-3[1/s])`, where the `mbd.diag` variable contains the diagonal of the bounding box of the geometry. The characteristic slip velocity should be small when compared to actual velocities in the joint. It is the velocity with which the joint slides even under sticking conditions.

- For the [Hinge Joint](#), enter a **Hinge radius** r .
- For the [Cylindrical Joint](#), enter a **Cylinder radius** r .
- For the [Screw Joint](#), enter a **Screw radius** r .
- For the [Ball Joint](#), enter a **Ball radius** r .

The radius is used for computing the friction moment in the joints that have a rotational degree of freedom.

For the [Planar Joint](#), select a **Friction model** — **Total force at center of joint** or **Distributed force on selected boundaries**. By distributing the friction forces, it is possible to obtain a moment caused by friction for a rotational relative motion. When **Distributed force on selected boundaries** is selected, a [Contact Area](#) subnode is added by default. In that subnode you make the selection of the boundaries on which the friction forces should be distributed.

NORMAL FORCE

In this section you supply details about how to compute the force acting on the joint, to which the friction loss is proportional.



For a detailed discussion of the effect of the settings for the normal force for each joint type, see [Theory for Friction in Joints](#)

The **Contribution from joint forces** check box is selected by default, which implies that the friction forces are computed based on normal forces determined from the forces acting the joint. If you click to clear the check box the friction force is independent of the forces acting on the joint, and the following parameter selection is not required.

Select an option from the **Normal force defined through** list — **Attachment reaction forces**, **Joint forces and moments**, or for the [Prismatic Joint](#), **Joint force in specified direction**.



For the [Ball Joint](#), no further selection is required.



The alternative **Attachment reaction forces** is not available if the joint is attached to a rigid domain or to an attachment in a Shell or Beam interface.

For **Joint forces and moments** enter values for the **Joint moment contribution factors** α_{11} (for the **Prismatic Joint** only), α_{12} , and α_{13} .

- For the **Prismatic Joint**, and if **Joint force in specified direction** is selected, enter values for each space coordinate for the **Direction of surface normal** \mathbf{e}_{n0} .

By selecting **Contribution from interference fit** check box you can define the normal forces by pressure or total force from an assumed interference fit.

Select an option from the **Normal force defined through** list — **Pressure** or **Total force**.

- For **Pressure** enter a value for p . Also enter a **Contact area** A .
- For **Total force** enter a value for F_n .

ADVANCED

Enter values or expressions for the:

- **Additional sliding resistance** $F_{f,r}$.
- **Maximum friction force** $F_{f,max}$. The default is Inf, which means that no upper limit is imposed on the friction force.



Theory for Friction in Joints

LOCATION IN USER INTERFACE

Context Menus

- Multibody Dynamics>Prismatic Joint>Friction**
- Multibody Dynamics>Hinge Joint>Friction**
- Multibody Dynamics>Cylindrical Joint>Friction**
- Multibody Dynamics>Screw Joint>Friction**
- Multibody Dynamics>Planar Joint>Friction**
- Multibody Dynamics>Ball Joint>Friction**

Ribbon

Physics tab with **Prismatic Joint**, **Hinge Joint**, **Cylindrical Joint**, **Screw Joint**, **Planar Joint**, or **Ball Joint** selected in the model tree:

Attributes>Friction

Contact Area

Select the boundaries to be used for distributing the friction forces in a [Planar Joint](#) when **Friction model** is chosen as **Distributed force on selected boundaries**.

LOCATION IN USER INTERFACE

This node is automatically added when **Distributed force on selected boundaries** is selected in the parent joint node. It cannot be added or removed manually.

Joint Axis

Use the **Joint Axis** node to select any edge in the geometry which is parallel to the axis of the joint.

This setting is available for the [Prismatic Joint](#), [Hinge Joint](#), [Cylindrical Joint](#), [Screw Joint](#), and [Planar Joint](#).



Joint Theory

LOCATION IN USER INTERFACE

This node is automatically added when **Select a parallel edge** is selected in the parent joint node. It cannot be added or removed manually.

Input Shaft Orientation

Select any edge in the geometry which is parallel to the ingoing shaft in a [Universal Joint](#).



Universal Joint Theory

LOCATION IN USER INTERFACE

This node is automatically added when **Select a parallel edge** is selected under **Input Shaft Orientation** in the parent **Universal Joint** node. It cannot be added or removed manually.

Output Shaft Orientation

Select any edge in the geometry which is parallel to the outgoing shaft in a [Universal Joint](#).



[Universal Joint Theory](#)

LOCATION IN USER INTERFACE

This node is automatically added when **Select a parallel edge** is selected under **Output Shaft Orientation** in the parent **Universal Joint** joint node. It cannot be added or removed manually.

Center of Joint Nodes



[Positioning the Joint Center](#)

CENTER OF JOINT: BOUNDARY

Select a set of boundaries whose centroid represents the center of the joint.

CENTER OF JOINT: EDGE

Select a set of edges whose centroid represents the center of the joint.

CENTER OF JOINT: POINT

Select a set of points whose centroid represents the center of the joint.

LOCATION IN USER INTERFACE

These nodes are automatically added when **Centroid of selected entities** is chosen in the parent joint node. The type of node generated depends in the setting of **Entity level**. They cannot be added or removed manually.

Source Point Nodes

SOURCE POINT: BOUNDARY

Select a set of boundaries whose centroid represents the source connection point.

SOURCE POINT: EDGE

Select a set of edges whose centroid represents the source connection point.

SOURCE POINT: POINT

Select a set of points whose centroid represents the source connection point.

LOCATION IN USER INTERFACE

When **Centroid of selected entities** is chosen under **Connection point** for the source attachment in the parent [Distance Joint](#), a subnode will be generated based on the setting of **Entity level**. They cannot be added or removed manually.

Destination Point Nodes

DESTINATION POINT: BOUNDARY

Select a set of boundaries whose centroid represents the destination connection point.

DESTINATION POINT: EDGE

Select a set of edges whose centroid represents the destination connection point.

DESTINATION POINT: POINT

Select a set of points whose centroid represents the destination connection point.

LOCATION IN USER INTERFACE

When **Centroid of selected entities** is chosen under **Connection point** for the destination attachment in the parent [Distance Joint](#), a subnode will be generated based on the setting of **Entity level**. They cannot be added or removed manually.

Source Filter

When you select domains in this node, the **Source** list in the parent node will only contain attachments and rigid domains that are part of the selection. The selection rules are:

- If any part of an active rigid domain is within the selection, that rigid domain will appear in the list.
- If any boundary of an active attachment is adjacent to the selected domains, that attachment will appear in the list.
- If the selection is such that no attachments or rigid domains satisfy the rules above, the **Source** list in the parent node will only contain the entry **None** (if the parent node is a joint), or **Select a connection point** (if the parent node is a Spring-Damper).
- If the selection is such that some attachments or rigid domains satisfy the rules above, the **None** entry in the **Source** list in the parent node will be removed, and one of the attachments or rigid domains will be selected as default. This means that if the selection is such that the **Source** list has only one unique item, this item is automatically selected.
- If the selection is empty, the behavior is as if **Use selection filter** were not selected.

LOCATION IN USER INTERFACE

This node is automatically created when the **Use selection filter** check box is selected in the **Attachment Selection** section in the parent node. The parent can be any joint node, or a [Spring-Damper](#) node. The **Source Filter** node cannot be added or removed manually.


Destination Filter

When you select domains in this node, the **Destination** list in the parent node will only contain attachments and rigid domains which are part of the selection. The selection rules are described in [Source Filter](#).

LOCATION IN USER INTERFACE

This node is automatically created when the **Use selection filter** check box is selected in the **Attachment Selection** section in the parent node. The parent can be any joint node, or a [Spring-Damper](#) node. The **Destination Filter** node cannot be added or removed manually.

Spring-Damper

Use a **Spring-Damper** () for 2D and 3D components to model — between two points — an elastic spring, a viscous damper, or both.

ATTACHMENT SELECTION

Select the two points between which the spring or damper is connected.

The settings are similar to the corresponding settings for the [Distance Joint](#) but applied to Spring-Damper. There is however one more alternative for both **Source** and **Destination**; **Select a connection point**. This is the default option, and then you do the actual selections in the **Source Point** and **Destination Point** subnodes. If you make a nondefault choice for **Source** or **Destination**, the corresponding selection subnode will be removed.

In large models, the list of available attachments can become very long. You can then create geometric filters to narrow down the search. When the **Use selection filter** check box is selected, two subnodes named [Source Filter](#) and [Destination Filter](#) are added to the **Spring-Damper** node. In these subnodes you can make graphic selections of the objects whose attachments should be shown in the **Source** and a **Destination** lists.

SPRING - DAMPER

Select a **Spring type** — **Spring constant** or **Force as function of extension**.

- For **Spring constant** enter a value for k .
- For **Force as function of extension** enter an expression for F_s . The expression must be a function of the extension of the spring. The built-in variable for the spring extension has the form `<physicsName>.<SpringNodeTag>.d1`, for example `mbd.spd1.d1`.

To add viscous damping in a dynamic analysis, enter a value or expression for the **Damping coefficient** c .

FREE LENGTH

The free length is the distance between the connection points when there is no force in the spring. Select an option from the list: **Specify initial extension** or **Specify free length**.

- For **Specify initial extension** enter a value for Δl_0 . The free length is computed as $l_f = l_0 - \Delta l_0$, where l_0 is the initial distance between the connection points.
- For **Specify free length** enter a value for l_f .

ACTIVATION CONDITION

Select a **Spring action** — **Bidirectional**, **Tension only**, or **Compression only**.

Select the **Deactivation condition** check box to enter a **Deactivation indicator expression** i_{dac} . The expression is treated as a boolean expression, so that when it evaluated to a nonzero value, the spring or damper is deactivated.

Select the **Permanently deactivate** check box if the spring is supposed to be removed permanently from the simulation once the deactivation condition is fulfilled for the first time.

As an example, if the spring should break at a certain extension, you can write an expression like `mbd.spd2.d1>0.12[m]`, and select the **Permanently deactivate** check box. If the check box is not selected, the spring would become active again when its connection points come close enough to each other.



Spring-Damper Theory

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Spring-Damper

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Multibody Dynamics>Spring-Damper

Source Point (for Spring-Damper)

Select a point, or a set of points, representing the source connection point for the [Spring-Damper](#). The main intent is that you should select a single connection point, but it is also possible to select several points. If you select several points, the spring attachment location is the average value of the coordinates of the selected points. The displacement is computed as the average of the displacements in the selected points, which for a flexible body may not be the same as the actual displacement in the averaged location. The spring force is distributed over the selected points.

LOCATION IN USER INTERFACE

When **Select a connection point** is chosen under **Source** for in the parent [Spring-Damper](#) node, this subnode will automatically be generated. It cannot be added or removed manually.


Destination Point (for Spring-Damper)

Select a point, or a set of points, representing the destination connection point for the [Spring-Damper](#). For details, see [Source Point \(for Spring-Damper\)](#).

LOCATION IN USER INTERFACE

When **Select a connection point** is chosen under **Destination** for in the parent [Spring-Damper](#) node, this subnode will automatically be generated. It cannot be added or removed manually.

Base Motion

Sometimes the components described by a multibody dynamics assembly are attached a common “ground”, which is not fixed in space but has a uniform movement. Use **Base Motion** () to specify the motion (displacement, velocity, or acceleration) of the environment to which the assembly is connected. The **Base Motion** node can then be referenced in the selection of source attachment for the joints and springs that connect to the base.

BASE MOTION

Select a **Base motion type** — **Displacement**, **Velocity**, or **Acceleration**.

Enter a value based on the selection:

- For **Displacement**, enter \mathbf{u}_b .
- For **Velocity**, enter \mathbf{v}_b .
- For **Acceleration**, enter \mathbf{a}_b .

The values are given in the coordinate system selected in the **Coordinate system selection** section.

INITIAL BASE DISPLACEMENT

This section is available when **Velocity** or **Acceleration** is selected as the **Base motion type**. Then enter a value for \mathbf{u}_{b0} to prescribe a nonzero initial value for the base

displacement. The values are given in the coordinate system selected in the **Coordinate system selection** section.

INITIAL BASE VELOCITY

This section is available when **Acceleration** is selected as the **Base motion type**. Then enter a value for \mathbf{v}_{b0} to prescribe a nonzero initial value for the base velocity. The values are given in the coordinate system selected in the **Coordinate system selection** section.

LOCATION IN USER INTERFACE

Context Menus


Multibody Dynamics>Base Motion

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Multibody Dynamics>Base Motion

Cam-Follower

Use the **Cam-Follower** () node to force a point of one moving body to maintain contact with a curve or surface of another moving body. This connection node is generic in nature and allows any point to follow a curve or surface, including a typical radial or cylindrical cam-follower connection used in various mechanisms.

The cam-follower connection models a simplified contact and establishes a higher kinematic pair between a cam and its follower.

BOUNDARY SELECTION, CAM

Select a set of boundaries which is being followed by a point called 'follower'. This option is available only when **Selection type** is set to **Boundary** in the **Cam** section. The boundary selection can be specified for 3D and 2D models.

EDGE SELECTION, CAM

Select a set of edges which is being followed by a point called “follower”. This option is available only when **Selection type** is set to **Edge** in the **Cam** section. The edge selection can be specified only for 3D models.

DOMAIN SELECTION, FOLLOWER

Select a set of domains such that the centroid of the selected entities follows the cam boundaries or edges with a certain optional offset. This option is available only when

Connection point is set to **Centroid of selected entities** and **Entity level** is set to **Domain** in the **Follower** section.

BOUNDARY SELECTION, FOLLOWER

Select a set of boundaries such that the centroid of the selected entities follows the cam boundaries or edges with a certain optional offset. This option is available only when **Connection point** is set to **Centroid of selected entities** and **Entity level** is set to **Boundary** in the **Follower** section.

EDGE SELECTION, FOLLOWER

Select a set of edges such that the centroid of the selected entities follows the cam boundaries or edges with a certain optional offset. This option is available only when **Connection point** is set to **Centroid of selected entities** and **Entity level** is set to **Edge** in the **Follower** section. The edge selection can be specified only for 3D models.

POINT SELECTION, FOLLOWER

Select a set of points such that the centroid of the selected entities follows the cam boundaries or edges with a certain optional offset. This option is available only when **Connection point** is set to **Centroid of selected entities** and **Entity level** is set to **Point** in the **Follower** section.

POINT SELECTION, FOLLOWER (SINGLETON)

Select a single point, such that the selected point follows the cam boundaries or edges with a certain optional offset. This option is available when **Connection point** is set to **Geometric Point** in the **Follower** section.

CAM

The cam selection can be defined on both rigid and elastic bodies. Thus, cam boundaries and edges can have any kind of rigid body motion as well as deformation. When defining a cam selection, it is also possible to use boundaries or edges that are defined outside the physics interface.

Select the **Selection type** — **Boundary** or **Edge**.

Enter a value of offset based on the selection:

- For **Boundary**, enter **Follower point offset from cam boundary**, X_{offset} , as a scalar value.
- For **Edge**, enter **Follower point offset from cam edge**, X_{offset} , as a vector.

FOLLOWER

A follower is defined through a point which follows the cam boundaries or edges with a certain optional offset. The follower point can be a part of rigid or elastic bodies. If there is a need for multiple points to follow the cam boundaries or edges, multiple instances the **Cam-Follower** feature should be used.

Select the **Connection point** — **Centroid of selected entities** or **Geometric point**.

When **Centroid of selected entities** is selected as a **Connection point**, specify the **Entity level** as **Domain**, **Boundary**, **Edge**, or **Point**.

CONNECTION FORCE

Select the **Connection force** — **Do not compute**, **Computed using weak constraints**, or **Computed using penalty method** in order to disable or enable the computation of the cam-follower connection force.

In case **Computed using penalty method** is selected, specify a value of **Penalty factor** (p_c) used for the connection constraint.

ACTIVATION CONDITION

Select **Connection state** — **Always active**, or **Conditionally active**.

When **Conditionally active** is selected, specify a value of the **Connection tolerance** (Δ) used in the connection constraint. This option can be used to deactivate the connection between the two bodies based on their spatial positions and the value of the connection tolerance.



LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Cam-Follower

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Multibody Dynamics>Cam-Follower

Spur Gear

Use the **Spur Gear** node to model a spur gear 2D and 3D. The gear is assumed to be rigid. Select one or more domains to make them a part of the gear.

A spur gear can be connected to the following gear types:

- Another [Spur Gear](#), or a [Helical Gear](#) through the [Gear Pair](#) node.
- A [Worm Gear](#) through the [Worm and Wheel](#) node.
- A [Spur Rack](#), or a [Helical Rack](#) through the [Rack and Pinion](#) node.

Any two gears or rack can be connected to each other only if they satisfy the [Gear Pair Compatibility Criteria](#).

The gear has the following subnodes to constraint the displacement and rotation, to apply forces and moments, and to add mass and moment of inertia:



These subnodes are described in the *Structural Mechanics Module User's Guide*:

- [Fixed Constraint \(Rigid Domain\)](#)
 - [Prescribed Displacement/Rotation](#)
 - [Applied Force \(Rigid Domain\)](#)
 - [Applied Moment \(Rigid Domain\)](#)
 - [Mass and Moment of Inertia \(Rigid Domain\)](#)
-

SKETCH

This section shows a sketch of a spur gear. Some of the inputs are also highlighted in the sketch.

GEAR PROPERTIES

Specify the properties of a spur gear in this section.

- Select the **Gear mesh** — **External** or **Internal**.
- Enter **Number of teeth** n .
- Enter **Pitch diameter** d_p .
- Enter **Pressure angle** α .

GEAR AXIS

The gear axis is defined as the axis of rotation of the gear. The gear axis is defined as the axis of rotation of the gear. To define the axis, choose **Specify direction** or **Select a parallel edge**.

- For **Specify direction**, enter a value or expression for \mathbf{e}_g . The default is (0, 0, 1). The direction is specified in the selected coordinate system.
- For **Select a parallel edge**, an edge can be selected on the [Gear Axis](#) subnode, which is added automatically. The vector from the first to last end of the edge is used to define the gear axis. Any edge in the model can be used. Select the **Reverse direction** check box to reverse the direction of the gear axis.

This section is only present in 3D. In 2D, the gear axis is assumed to be the out-of-plane direction.

DENSITY

The default **Density** ρ is taken **From material**. In this case, the material assignment for the domain supplies the mass density. For **User defined** enter another value or expression.



The density is needed for dynamic analysis. It is also used when computing mass forces for gravitational or rotating frame loads, and when computing mass properties ([Computing Mass Properties](#)).

CENTER OF ROTATION

Select a **Center of Rotation** — **Center of mass**, **Centroid of selected entities**, or **User defined**. The center of rotation is the point about which the rotation of the gear is interpreted.

- For **Center of mass**, the center of rotation is taken as the center of mass of the gear.
- For **Centroid of selected entities** select an **Entity level** — **Boundary**, **Edge**, or **Point**. The available choices depend on geometrical dimension. The center of rotation is located at the centroid of the selected entities, which do not need to be related to

rigid domain itself. As a special case, you can select a single point and thus use that point as center of rotation.



Once chosen, a default **Center of Rotation: Boundary**, **Center of Rotation: Edge**, or **Center of Rotation: Point** subnode is automatically added.

- For **User defined**, enter the **Global coordinates of center of rotation**, \mathbf{X}_e , in the table.

Select the **Offset** check box to add an optional offset vector to the definition of the center of rotation. Enter values for the offset vector $\mathbf{X}_{\text{offset}}$.

The center of rotation used is the sum of the vector obtained from any of the input methods and the offset vector.

$$\mathbf{X}_P = \mathbf{X}_{P,\text{input}} + \mathbf{X}_{\text{offset}}$$

INITIAL VALUES

Select **From physics interface node** or **Locally defined**.

- When **From physics interface node** is selected, initial values for rigid body displacement, rotation, and velocities are inherited from the physics interface level.
- When **Locally defined** is selected, an **Initial Values** subnode is automatically added, in which you can initialize the rigid domain degrees of freedom.

Select a **Consistent initialization** — **Default** or **Force initial values** to enforce that the exact values given in this node should be maintained through the consistent initialization process. The default when the given initial values are inconsistent is to make them adapt to each other by adjusting all initial values. For **Force initial values** click to select the applicable check box or boxes — **Translation along first axis**, **Translation along second**

axis, **Translation along third axis**, and **Total rotation**. Only the selected degrees of freedom have the initial values fixed during the consistent initialization process..



- The optional subnode is described in [Initial Values \(Rigid Domain\)](#).
- For details about the input expressions at the physics interface level, see [Initial Values](#).
- See [Rigid Domain Model](#) in the *Structural Mechanics Module User's Guide* for more information.
- For details about the initialization process, see [Initialization in Time-Dependent Studies](#).



[Spur Gear Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Spur Gear

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Spur Gear

Helical Gear

Use the **Helical Gear** node to model a helical gear in 2D and 3D. The gear is assumed to be rigid. Select one or more domains to make them a part of the gear.

A helical gear can be connected to the following gear types:

- Another [Helical Gear](#), or a [Spur Gear](#) through the [Gear Pair](#) node.
- A [Worm Gear](#) through the [Worm and Wheel](#) node.
- A [Helical Rack](#), or a [Spur Rack](#) through the [Rack and Pinion](#) node.

Any two gears or rack can be connected to each other only if they satisfy the [Gear Pair Compatibility Criteria](#).

GEAR PROPERTIES

Specify the properties of a helical gear in this section.

- Select the **Gear mesh** — **External** or **Internal**.
- Enter **Number of teeth** n .
- Enter **Pitch diameter** d_p .
- Enter **Pressure angle** α .
- Enter **Helix angle** β .



See [Spur Gear](#) for the list of subnodes and the following settings — **Sketch**, **Gear Axis**, **Density**, **Center of Rotation**, and **Initial Values**.



[Helical Gear Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Helical Gear

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Helical Gear

Bevel Gear

Use the **Bevel Gear** node to model a bevel gear in 3D. The gear is assumed to be rigid. Select one or more domains to make them a part of the gear.

A bevel gear can be connected to another [Bevel Gear](#) through the [Gear Pair](#) node. Two bevel gears can be connected to each other only if they satisfy the [Gear Pair Compatibility Criteria](#).

GEAR PROPERTIES

Specify the properties of a bevel gear in this section.

- Enter **Number of teeth** n .
- Enter **Pitch diameter** d_p .

- Enter **Pressure angle** α .
- Enter **Cone angle** γ .



See [Spur Gear](#) for the list of subnodes and the following settings — **Sketch, Gear Axis, Density, Center of Rotation, and Initial Values.**



[Bevel Gear Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Bevel Gear

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Bevel Gear

Worm Gear

Use this node to model a **Worm Gear** in 3D. The gear is assumed to be rigid. Select one or more domains to make them a part of a gear.

A worm gear can be connected to a [Helical Gear](#), or a [Spur Gear](#) through the [Worm and Wheel](#) node. A worm can be connected to a wheel only if they satisfy the [Gear Pair Compatibility Criteria](#).

GEAR PROPERTIES

Specify the properties of a worm gear in this section.

- Enter **Number of starts** n .
- Enter **Pitch diameter** d_p .
- Enter **Pressure angle** α .

- Enter **Lead angle** ϕ .
- Select the **Handedness** — **Right**, or **Left**..



See [Spur Gear](#) for the list of subnodes and the following settings — **Sketch**, **Gear Axis**, **Density**, **Center of Rotation**, and **Initial Values**.



The default value of gear axis for a worm gear is (1, 0, 0).



[Worm Gear Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Worm Gear

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Worm Gear

Spur Rack

Use this node to model a **Spur Rack** in 2D and 3D. The rack is assumed to be rigid. Select one or more domains to make them a part of the rack.

A spur rack can be connected to a [Spur Gear](#), or a [Helical Gear](#) through the [Rack and Pinion](#) node. A rack can be connected to a pinion only if they satisfy the [Gear Pair Compatibility Criteria](#).

RACK PROPERTIES

Specify the properties of a rack in this section.

- Enter **Pitch** p .
- Enter **Pressure angle** α .

RACK AXIS

The rack axis is the axis of translation of the rack. To define the axis, choose — **Specify direction**, or **Select a parallel edge**.

- For **Specify direction** enter a value or expression for \mathbf{e}_r . The default is (1, 0, 0). The direction is specified in the selected coordinate system.
- For **Select a parallel edge** an edge can be selected on the [Rack Axis](#) subnode, which is added automatically. The vector from the first to the last end of the edge is used to define the rack axis. Any edge in the model can be used. Select the **Reverse direction** check box to reverse the direction of the rack axis..



See [Spur Gear](#) for the list of subnodes and the following settings — **Sketch**, **Density**, **Center of Rotation**, and **Initial Values**.



[Spur Rack Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Spur Rack

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Spur Rack

Helical Rack

Use this node to model a **Helical Rack** in 2D and 3D. The rack is assumed to be rigid. Select one or more domains to make them a part of the rack.

A helical rack can be connected to a [Spur Gear](#), or a [Helical Gear](#) through the [Rack and Pinion](#) node. A rack can be connected to a pinion only if they satisfy the [Gear Pair Compatibility Criteria](#).

RACK PROPERTIES

Specify the properties of a rack in this section.

- Enter **Pitch** p .

- Enter **Pressure angle** α .
- Enter **Helix angle** β .



See [Spur Rack](#) for the list of subnodes and the following settings — **Sketch, Rack Axis, Density, Center of Rotation, and Initial Values.**



[Helical Rack Theory](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gears>Helical Rack

Ribbon

Physics tab with **Multibody Dynamics** selected:

Domains>Gears>Helical Rack

Gear Axis

Use the **Gear Axis** node to select any edge in the geometry that is parallel to the axis of rotation of the gear.

This setting is available for the [Spur Gear](#), [Helical Gear](#), [Bevel Gear](#), and [Worm Gear](#) nodes.

LOCATION IN USER INTERFACE

This node is automatically added when **Select a parallel edge** is selected in the parent gear node. It cannot be added or removed manually.

Rack Axis

Use the **Rack Axis** node to select any edge in the geometry which is parallel to the axis of translation of the rack.

This setting is available for the [Spur Rack](#) and [Helical Rack](#).

LOCATION IN USER INTERFACE

This node is automatically added when **Select a parallel edge** is selected in the parent rack node. It cannot be added or removed manually.

Gear Pair

Use the **Gear Pair** node for 2D and 3D components to create a connection between gears of the types [Spur Gear](#), [Helical Gear](#), or [Bevel Gear](#) in such a way that, at the contact point, they have no relative motion along the line of action. The remaining displacements and rotations of both the gears are independent of each other.

If the **Line contact** option is chosen in the **Contact model**, one additional constraint is added at the contact point to restrict the relative rotation about a line joining the two gear centers.

When **Friction** is included for a gear pair, frictional forces in the plane perpendicular to the line of action are added on both the gears.

SKETCH

This section shows a sketch of a gear pair. The sketch highlights the coordinate system attached to both the gears, the coordinate system attached to the gear tooth, the line of action, and other relevant input parameters.

GEAR SELECTION

Select **Wheel** and **Pinion** from the list of gears. You can select any [Spur Gear](#), [Helical Gear](#), or [Bevel Gear](#) node that is defined for the model.

LINE OF ACTION

Define the line of action by specifying **Obtained by rotation of tangent in — Clockwise direction**, or **Counterclockwise direction**.

Select the **Clockwise direction** option in the following cases:

- **Wheel** is the driver and the **Wheel** has positive rotation about its own axis.
- **Pinion** is the driver and the **Wheel** has negative rotation about its own axis.

Select the **Counterclockwise direction** option in the following cases:

- **Pinion** is the driver and the **Wheel** has positive rotation about its own axis.
- **Wheel** is the driver and the **Wheel** has negative rotation about its own axis.

The gear pair sketch in the **Sketch** section is updated based on the selection and shows the line of action for both cases.

CONTACT MODEL

Define the contact model for a gear pair in this section.

- Select the **Configuration** — **Parallel or intersecting** or **Neither parallel nor intersecting**. The **Configuration** is always set to **Parallel or intersecting** for spur and bevel gears. For helical gears, you can select the **Configuration** as **Parallel or intersecting** (for parallel axis helical gears) or **Neither parallel nor intersecting** (for crossed axis helical gears).
- For **Parallel or intersecting**, select the **Contact model** — **Point contact**, or **Line contact**. For **Neither parallel nor intersecting** the **Contact model** is always set to **Point contact**.
- For **Parallel or intersecting**, enter **Contact point offset from pinion center** z_{pn} . For **Neither parallel nor intersecting** it is automatically computed.
- For **Line contact**, enter **Working width of pinion** w_{pn} .



In 2D, there are no settings for selecting the **Contact Model**. The **Configuration** is always **Parallel** and the **Contact model** is always set to the **Point contact**.

GEAR PAIR PROPERTIES

Define the advanced properties for a gear pair in this section.

- Select **Include gear elasticity** to enter the gear mesh stiffness, mesh damping, and the contact ratio. These properties can be entered in the [Gear Elasticity](#) subnode, which is added automatically. By default, this option is not selected.
- Select **Include transmission error** to enter the static transmission error for individual gears or for the gear pair. These can be entered in the [Transmission Error](#) subnode, which is added automatically. By default, this option is not selected.
- Select **Include backlash** to enter backlash in a gear pair. This can be entered in the [Backlash](#) subnode, which is added automatically. By default, this option is not selected.
- Select **Include friction** to add friction at the contact point. Enter frictional properties in the [Friction \(Gear\)](#) subnode, which is added automatically. By default, this option is not selected.

CONTACT FORCE COMPUTATION

Select an option from the list to use the method of computation for the forces at the contact point — **Computed using weak constraints**, or **Computed using penalty method**. Alternatively, you can also choose **Do not compute**.

For **Computed using penalty method**, enter **Penalty factor** p_c . The default value is $((1[1/ms])^2) * mbd.grp1.me$. The exact naming of the variable depends on the tags of the nodes in the model tree.



Theory for Gear Pairs

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gear Pairs>Gear Pair

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Gear Pairs>Gear Pair

Worm and Wheel

Use the **Worm and Wheel** node for 3D components to create a connection between a [Worm Gear](#) and a [Helical Gear](#) or a [Spur Gear](#) in such a way that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of both the gears are independent of each other.

When **Friction** is included on a **Worm and Wheel** node, frictional forces in the plane perpendicular to the line of action are added on both the gears.

GEAR SELECTION

Select **Worm** and **Wheel** from the list of gears. You can select any [Worm Gear](#) in the **Worm** field and any [Helical Gear](#) or [Spur Gear](#) in the **Wheel** field.



See [Gear Pair](#) for the following settings — **Sketch**, **Line of Action**, **Gear Pair Properties**, and **Contact Force Computation**.



In this node, there are no settings for **Contact Model**. The **Configuration** is always **Neither parallel nor intersecting** and the **Contact model** is always set to the **Point contact**.



In this node, the **Include friction** option is selected by default, and a **Friction** subnode is thus added.



[Theory for Worm and Wheel](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gear Pairs>Worm and Wheel

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Gear Pairs>Worm and Wheel

Rack and Pinion

Use the **Rack and Pinion** node for 2D and 3D components to create a connection between a [Spur Rack](#) and a [Spur Gear](#), or between a [Helical Rack](#) and a [Helical Gear](#) in such a way that at the contact point they have no relative motion along the line of action. The remaining displacements and rotations of the rack and pinion are independent of each other.

If the **Line contact** option is chosen in the **Contact model**, one additional constraint is added at the contact point to restrict the relative rotation about a line joining the rack and pinion centers.

When **Friction** is included on a rack and pinion node, frictional forces in the plane perpendicular to the line of action are added on the rack and pinion.

GEAR SELECTION

Select **Rack** and **Pinion** from the list of racks and gears. You can select any [Spur Rack](#), or [Helical Rack](#) in the **Rack** field and any [Spur Gear](#), or [Helical Gear](#) in the **Pinion** field that is defined for the model.



See [Gear Pair](#) for the following settings — **Sketch**, **Line of Action**, **Contact Model**, **Gear Pair Properties**, and **Contact Force Computation**.



In this node, there is no setting for **Configuration** in the **Contact Model** section since it is possible to have either **Point Contact** or **Line Contact** independent of the configuration,



[Theory for Rack and Pinion](#)

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Gear Pairs>Rack and Pinion

Ribbon

Physics tab with **Multibody Dynamics** selected:

Global>Gear Pairs>Rack and Pinion

Gear Elasticity

The **Gear Elasticity** subnode is automatically added to a [Gear Pair](#), [Worm and Wheel](#), or [Rack and Pinion](#) node when **Include gear elasticity** is selected under **Gear Properties**. In this node, you can enter the elastic properties of a gear pair, such as gear mesh stiffness, mesh damping, and contact ratio.

The contents of the **Gear Elasticity** settings adapt to the type of gear pair where it is used.

SKETCH

This section shows a sketch of the contact point of gear pair in the gear tooth normal plane. The sketch highlights the line of action along which stiffness and damping are interpreted.

MESH STIFFNESS

Select the mesh stiffness method **Specify** — **Tooth stiffness of individual gears** or **Total stiffness of gear pair**.

For **Tooth stiffness of individual gears**, enter the following:

- When the parent node is **Gear Pair**, enter **Wheel tooth stiffness as function of mesh cycle**, $k_{t,wh}$, and **Pinion tooth stiffness as function of mesh cycle**, $k_{t,pn}$. The default expressions are $(0[N/m]) * mbd.grp1.thm_wh$ and $(0[N/m]) * mbd.grp1.thm_pn$ respectively. The exact naming of the variables depends on the tags of the nodes in the model tree.
- When the parent node is **Worm and Wheel**, enter **Worm tooth stiffness as function of mesh cycle**, $k_{t,wm}$, and **Wheel tooth stiffness as function of mesh cycle**, $k_{t,wh}$. The default expressions are $(0[N/m]) * mbd.grp1.thm_wm$ and $(0[N/m]) * mbd.grp1.thm_wh$ respectively. The exact naming of the variables depends on the tags of the nodes in the model tree.
- When the parent node is **Rack and Pinion**, enter **Rack tooth stiffness as function of mesh cycle**, $k_{t,rk}$, and **Pinion tooth stiffness as function of mesh cycle**, $k_{t,pn}$. The default expressions are $(0[N/m^2]) * mbd.grp1.um_rk$ and $(0[N/m]) * mbd.grp1.thm_pn$ respectively. The exact naming of the variables depends on the tags of the nodes in the model tree.
- Specify **Contact ratio in mesh cycle** — **Constant** or **Varying**.
- For **Constant** contact ratio, select **Contact ratio** — **1**, **2**, or **3**.
- For **Varying** contact ratio, select **Maximum contact ratio** — **2**, or **3**. Also enter the **Next tooth engagement position in mesh cycle**, ζ . The value of ζ must vary between 1/2 and 1 when **Maximum contact ratio** is set to **2** and between 1/3 to 1/2 when **Maximum contact ratio** is set to **3**.

For **Total stiffness of gear pair**, enter the following:

- In case the parent node is **Gear Pair**, select the **Total stiffness as function of** — **Wheel mesh cycle**, **Wheel full revolution**, **Pinion mesh cycle**, or **Pinion full revolution**.
- In case the parent node is **Worm and Wheel**, select the **Total stiffness as function of** — **Worm mesh cycle**, **Worm full revolution**, **Wheel mesh cycle**, or **Wheel full revolution**.
- In case the parent node is **Rack and Pinion**, select the **Total stiffness as function of** — **Rack mesh cycle**, **Pinion mesh cycle**, or **Pinion full revolution**.
- Enter **Total stiffness**, k_g . The default expression of total stiffness depends on the selection in previous input; for example, it is $(0[N/m]) * mbd.grp1.thm_wh$ when

Wheel mesh cycle is selected. The exact naming of the variable depends on the tags of the nodes in the model tree.

MESH DAMPING

Enter **Mesh damping**, c_g . The mesh damping can be specified as proportional to the mesh stiffness. The default expression is $(0[\text{s}]) * \text{mbd.grp1.kg}$. The exact naming of the variable depends on the tags of the nodes in the model tree.



Theory for Gear Elasticity

LOCATION IN USER INTERFACE

This node is automatically added when **Include gear elasticity** is selected in the parent [Gear Pair](#), [Worm and Wheel](#), and [Rack and Pinion](#) nodes. It cannot be added or removed manually.

Transmission Error

The **Transmission Error** subnode is automatically added to a [Gear Pair](#), [Worm and Wheel](#), or [Rack and Pinion](#) node when **Include transmission error** is selected under **Gear Properties**. The static transmission error for individual gears or for a gear pair can be entered in this node.

The contents of the **Transmission Error** settings adapt to the type of gear pair where it is used.

SKETCH

This section shows a sketch of the contact point of a gear pair in the gear tooth normal plane. The sketch highlights the line of action along which transmission error is interpreted.

TRANSMISSION ERROR

Select the transmission error method **Specify** — **Transmission error of individual gears**, or **Transmission error of gear pair**.

For **Transmission error of individual gears** enter the followings:

- When the parent node is [Gear Pair](#), select the **Wheel transmission error as function of** — **Mesh cycle**, or **Full revolution** and enter **Wheel transmission error**, e_{wh} . Similarly,

select the **Pinion transmission error as function of** — **Mesh cycle**, or **Full revolution** and enter **Pinion transmission error**, e_{pn} .

- When the parent node is **Worm and Wheel**, select the **Worm transmission error as function of** — **Mesh cycle**, or **Full revolution** and enter **Worm transmission error**, e_{wm} . Similarly, select the **Wheel transmission error as function of** — **Mesh cycle**, or **Full revolution** and enter **Wheel transmission error**, e_{wh} .
- When the parent node is **Rack and Pinion**, enter **Rack transmission error** e_{rk} as a function of **Mesh cycle**. Similarly select the **Pinion transmission error as function of** — **Mesh cycle**, or **Full revolution** and enter **Pinion transmission error**, e_{pn} .

The default expression for transmission errors depends on the selection in previous input (for example, the default expression of **Wheel transmission error** is $(O[m])*mbd.grp1.thm_wh$ when **Mesh cycle** is selected). The exact naming of the variable depends on the tags of the nodes in the model tree.

For **Transmission error of gear pair** enter the following:

- When the parent node is **Gear Pair**, select the **Transmission error as function of** — **Wheel mesh cycle**, **Wheel full revolution**, **Pinion mesh cycle**, or **Pinion full revolution**.
- When the parent node is **Worm and Wheel**, select the **Transmission error as function of** — **Worm mesh cycle**, **Worm full revolution**, **Wheel mesh cycle**, or **Wheel full revolution**.
- When the parent node is **Rack and Pinion**, select the **Transmission error as function of** — **Rack mesh cycle**, **Pinion mesh cycle**, or **Pinion full revolution**.

Enter **Total transmission error**, e_t . The default expression of total transmission error depends on the selection in previous input (for example, it is $(O[m])*mbd.grp1.thm_wh$ when **Wheel mesh cycle** is selected). The exact naming of the variable depends on the tags of the nodes in the model tree.



Theory for Transmission Error

LOCATION IN USER INTERFACE

This node is automatically added when **Include transmission error** is selected in the parent **Gear Pair**, **Worm and Wheel**, and **Rack and Pinion** nodes. It cannot be added or removed manually.

Backlash

The **Backlash** subnode is automatically added to a [Gear Pair](#), [Worm and Wheel](#), or [Rack and Pinion](#) node when **Include backlash** is selected under **Gear Properties**.



Backlash can only be specified on a gear pair in a time-dependent analysis.

SKETCH

This section shows a sketch of the contact point of gear pair in the gear tooth normal plane. The sketch highlights the direction in which backlash is interpreted.

BACKLASH

- Enter **Backlash** b_1 .
- Enter **Penalty factor** p_b . The default expression is $((1[1/ms])^2) * mbd.grp1.Ie$. The exact naming of the variable depends on the tags of the nodes in the model tree.

ADVANCED

- Select **Manual control on time stepping** to restrict the maximum time step taken by the solver when there is finite penetration.
- Enter **Maximum allowable time step** Δt_{max} . The default expression is $(\pi/30) * (\text{sqrt}(mbd.grp1.Ie/mbd.grp1.pb))$. The exact naming of the variables depends on the tags of the nodes in the model tree.



[Theory for Backlash](#)

LOCATION IN USER INTERFACE

This node is automatically added when **Include backlash** is selected in the parent [Gear Pair](#), [Worm and Wheel](#), and [Rack and Pinion](#) nodes. It cannot be added or removed manually.

Friction (Gear)

The **Friction** subnode is automatically added to a [Gear Pair](#), [Worm and Wheel](#), or [Rack and Pinion](#) node when **Include friction** is selected under **Gear Properties**. Use this node to model friction losses on a gear pair.



Friction can only be applied to a gear pair in a time-dependent analysis.

FRICITION

- Enter a **Frictional coefficient** μ .
- Enter a **Characteristic slip velocity** v_0 . The default is `rotsld.diag*(1e-3[1/s])`, where the `mbd.diag` variable contains the diagonal of the bounding box of the geometry. The characteristic slip velocity should be small when compared to the actual slip velocity at the contact point. It is the velocity with which the contact surfaces slide, even under sticking conditions.

NORMAL FORCE

- Specify **Normal force defined through** — **Contact force** or **User defined force**. For **Contact force** option, it is required that contact force is computed on the parent **Gear Pair** node.
- For the **User defined force** option, enter **Normal force** N .

ADVANCED

Enter values or expressions for the:

- **Additional sliding resistance** $F_{f,r}$.
 - **Maximum friction force** $F_{f,max}$. The default is `Inf`, which means that no upper limit is imposed on the friction force.
-



[Theory for Gear Friction](#)

LOCATION IN USER INTERFACE

This node is automatically added when **Include friction** is selected in the parent [Gear Pair](#), [Worm and Wheel](#), and [Rack and Pinion](#) nodes. It cannot be added or removed manually.

Radial Roller Bearing

Add a **Radial Roller Bearing** node to model a surface of the component supported by a roller bearing. The selected boundaries should form a cylindrical surface representing the surface connected with the inner race of the bearing. The forces on the rollers are computed using the Hertz contact theory. The built-in variable for the roller force distribution has the form `<physicsName>.<RollerBearingTag>.fb[x,y,z]`, for example `mbd.rrb1.fbx`.

BEARING ORIENTATION

Use the settings in this section to define the local *y*-direction of the bearing. Subsequent vector and matrix inputs are specified in the local directions specified here.

Enter the **Bearing orientation vector defining the local *y*-direction**. The direction given will be adjusted so that it is orthogonal to the rotor axis. The default value is the global *y*-axis.

You can further modify the *y*-axis orientation by entering an angle for the **Rotation of the orientation vector around the bearing axis**. The direction inferred from the previous setting will be rotated counterclockwise around the rotor axis.

GEOMETRIC PROPERTIES

Select a **Bearing Type** — **Deep groove ball bearing**, **Angular contact ball bearing**, **Self aligning ball bearing**, **Spherical roller bearing**, **Cylindrical roller bearing**, or **Tapered roller bearing**. Then go to the relevant section below to continue defining the properties.

Deep Groove Ball Bearing

- Select either **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.
- For a **Double row** bearing, enter the **Axial distance between the roller centers**.
- Enter the renaming geometric properties **Number of balls**; **Ball diameter**; **Radial clearance**; **Pitch diameter**; **Contour radius, inner race**; and **Contour radius, outer race** to define the bearing geometry.

Angular Contact Ball Bearing

- Select either **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.

- For a **Double row** bearing, enter the **Axial distance between the roller centers**.
- Enter the remaining geometric properties **Number of balls**; **Ball diameter**; **Radial clearance**; **Pitch diameter**; **Contour radius, inner race**; **Contour radius, outer race**; and **Initial contact angle** to define the bearing geometry.

Self Aligning Ball Bearing

- Select either **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.
- Enter the geometric properties **Number of balls**; **Ball diameter**; **Radial clearance**; **Pitch diameter**; **Contour radius, inner race**; and **Contour radius, outer race**.
- For a **Double row** bearing, enter also **Initial contact angle**.

Spherical Roller Bearing

- Select either **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.
- For a **Double row** bearing, enter the **Axial distance between the roller centers** and **Initial contact angle**.
- Enter the remaining geometric properties **Number of rollers**; **Roller diameter**; **Radial clearance**; **Pitch diameter**; **Contour radius, inner race**; and **Contour radius, outer race** to define the bearing geometry.

Cylindrical Roller Bearing

- Select either of **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.
- For a **Double row** bearing, enter the **Axial distance between the roller centers**.
- Enter the remaining geometric properties **Number of rollers**, **Effective length of the roller**, **Radial clearance**, and **Pitch diameter** to define the bearing geometry.

Tapered Roller Bearing

- Select either of **Single row** or **Double row** to specify if the bearing consists of single or double rows of rollers.
- For **Single row** bearing, enter the **Axial offset of roller center**.
- For **Double row** bearing enter the **Axial distance between the roller centers**.
- Enter the remaining geometric properties **Number of rollers**, **Roller diameter**, **Effective length of the roller**, **Inner race clearance**, **Outer race clearance**, **Flange clearance**, **Pitch diameter**, **Cap radius**, **Semi cone angle**, **Roller axis orientation**, **Cap angle**, and **Flange contact angle** to define the bearing geometry.

MATERIAL PROPERTIES

- Enter the material properties **Young's modulus, inner race** and **Poisson's ratio, inner race** to define the elastic properties of the inner race in the bearing.
- Select **Outer race material** — **Same as inner race** or **User defined**. If **User defined** is selected, enter the values of **Young's modulus, outer race** and **Poisson's ratio, outer race**.
- Finally select **Roller material** — **Same as inner race** or **User defined**. If **User defined** is selected, enter the values of the **Young's modulus, roller/ball** and **Poisson's ratio, roller/ball**.

FOUNDATION PROPERTIES

Select one of the **Fixed**, **Moving**, or **Flexible** foundation options. In the **Moving** and **Flexible** cases, the subnodes [Moving Foundation](#) or [Flexible Foundation](#) are automatically added.



To know the details of contact force computation in different roller bearings, see the theory for [Radial Roller Bearing](#) in the *Rotordynamics Module's User Guide*.

LOCATION IN USER INTERFACE

Context Menus

Multibody Dynamics>Radial Roller Bearing

Ribbon

Physics tab with **Multibody Dynamics** selected:

Boundaries>Multibody Dynamics>Radial Roller Bearing

Flexible Foundation

The **Flexible Foundation** node is added automatically when you set **Foundation properties** to **Flexible** on the parent **Radial Roller Bearing** node. In this node, you describe the flexibility of the foundation on which the bearing is mounted.

FOUNDATION PROPERTIES

Enter the values of the translational and rotational stiffness k_u and k_θ of the foundation. Select the **Translational-rotational coupling** check box to also incorporate

the effect of coupled stiffness between the translational and rotational motion of the foundation. If selected, enter the values of the coupling stiffnesses, $k_{u\theta}$ and $k_{\theta u}$.

If you want to perform a dynamic analysis, you can optionally also enter the values of the translational and rotational damping, c_u and c_θ , of the foundation. Select the **Translational-rotational coupling** check box to also incorporate the effect of coupled damping constants between the translational and angular velocity of the foundation. If selected, enter the values of the coupling damping constants $c_{u\theta}$ and $c_{\theta u}$.

LOCATION IN USER INTERFACE

This node is automatically added when **Foundation Properties** is **Flexible** in the parent bearing node. It cannot be added or removed manually.

Moving Foundation

The **Moving Foundation** node is added automatically when you set **Foundation properties** to **Moving** on the parent **Radial Roller Bearing** node. In this node, you prescribe the motion of the foundation on which the bearing is mounted.


FOUNDATION PROPERTIES

Enter the values of the **Foundation displacement**, \mathbf{u}_f , and **Foundation rotation**, θ_f , at the center of the bearing.

LOCATION IN USER INTERFACE

This node is automatically added when **Foundation Properties** is **Moving** in the parent bearing node. It cannot be added or removed manually.



Lumped Mechanical System

This chapter describes the Lumped Mechanical System interface, which is found under the **Structural Mechanics** branch () when adding a physics interface.

In this chapter:

- [The Lumped Mechanical System Interface](#)
- [Theory for the Lumped Mechanical System Interface](#)

The Lumped Mechanical System Interface

The **Lumped Mechanical System (lms)** interface () found under the **Structural Mechanics** branch () when adding a physics interface, is intended to model displacements and forces in a mechanical system using a circuit representation with lumped components such as masses, springs, and dampers. In addition to the displacement and force sources in the system, terminal nodes are used to define forces, displacements, velocities, accelerations, or impedances as boundary conditions. The physics interface supports stationary, frequency-domain, eigenfrequency and time-domain analysis. The models created with the Lumped Mechanical System interface can also include connections to distributed finite element models.

When the **Lumped Mechanical System** interface is added, one default node is also added to the **Model Builder — Fixed Node** (a boundary condition which acts as “ground”). Then, from the **Physics** toolbar, add more features in the model builder. You can also right-click **Lumped Mechanical System** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `lms`.

SYSTEM SETTINGS

Specify the following in this section:

- **Create unique nodes for new components**
- **Check for hanging nodes before solving**

When **Create unique nodes for new components** is selected, newly added components will be assigned unused node names. The components will be disconnected from the rest of the system and the nodes should be updated to reflect the actual system

connections. When this setting is deselected, new components will be connected to the lowest-numbered nodes, starting from 0.

When **Check for hanging nodes before solving** is selected, a check for hanging (not connected) nodes is performed before solving the equations. An error message is displayed if one or more hanging nodes are present in the system.

GRAVITY

Select **Include gravity** and specify **Acceleration due to gravity** (g) in order to add gravity force on all the mass components. It is possible to exclude the gravity force contribution from a selected mass component at the physics feature level.

DEPENDENT VARIABLES

Every feature of the physics interface adds certain ODE dependent variables in the form of displacement, force, velocity, and others. All the ODE dependent variables are grouped in the following ODE fields:

- **Displacements** (default field name: `displacements`)
- **Forces** (default field name: `forces`)
- **Velocities** (default field name: `velocities`)
- **Other** (default field name: `other`)

DISCRETIZATION

All the dependent variables in this physics interface are global and do not need any spatial discretization.

Nodes for the Lumped Mechanical System Interface

The **Lumped Mechanical System** has only global nodes. They are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking then main physics interface node to access the context menu (all users).



Nodes in the lumped mechanical system should not be confused with nodes in the Model Builder tree of the COMSOL Multiphysics software. The system node names are not restricted to numerical values but can contain alphanumeric characters.



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.


The following nodes are available from the **Physics** toolbar, **Global** menu or from by right-clicking the **Lumped Mechanical System** interface node.

- [Damper](#)
- [Displacement Source](#)
- [External Source](#)
- [Force Source](#)
- [Impedance](#)
- [Mass](#)
- [Spring](#)
- [Subsystem Definition](#)
- [Subsystem Instance](#)

The following nodes are available from the **Physics** toolbar, **Global** menu or from by selecting from the **Terminals** subnode after right-clicking the **Lumped Mechanical System** interface node.

- [Acceleration Node](#)
- [Displacement Node](#)
- [Fixed Node](#)
- [Force Node](#)
- [Free Node](#)
- [Impedance Node](#)
- [Velocity Node](#)

Mass

The **Mass** () feature connects a lumped mass between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the mass. The prefix is M.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the mass.

COMPONENT PARAMETERS

Specify the value of the **Mass**, m .

INITIAL VALUES

Enter the values of **Initial displacement** (u) and **Initial velocity** ($\frac{\partial u}{\partial t}$). The initial values are used in a time dependent analysis.

GRAVITY

Select the **Exclude contribution** check box in order to exclude the contribution of gravity force.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following quantities in the default plots or evaluations:

- **Force**
- **Displacement**
- **Velocity**
- **Acceleration**



Theory for Mass

Context Menus


Lumped Mechanical System>Mass

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Mass

Spring

The **Spring** () feature connects a linear spring between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the spring. The prefix is K.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the spring.

COMPONENT PARAMETERS

Specify the value of the **Spring Constant**, k , and the **Predeformation**, u_0 .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**



Theory for Spring

Context Menus


Lumped Mechanical System>Spring

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Spring

Damper

The **Damper** () feature connects a viscous damper between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the damper. The prefix is C.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the damper.

COMPONENT PARAMETERS

Specify the value of the **Damping coefficient**, c .

INITIAL VALUES

Enter the values of **Initial displacement**, u . The initial values are used in a time dependent analysis.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**
- **Velocity**



Theory for Damper

Context Menus


Lumped Mechanical System>Damper

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Damper

Impedance

The **Impedance** () feature connects a component, defined through an impedance matrix, between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the impedance. The prefix is **Z**.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the impedance.

COMPONENT PARAMETERS

Specify the values of the elements of the **Impedance** matrix, **Z**.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**
- **Velocity**



Theory for Impedance

Context Menus


Lumped Mechanical System>Impedance

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Impedance

Displacement Source

The **Displacement Source** () feature connects a displacement source between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the displacement source. The prefix is U.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the displacement source.

COMPONENT PARAMETERS

Specify the values of **Displacement**, u_{src} , and **Phase**, ϕ .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**



Theory for Displacement Source

Context Menus


Lumped Mechanical System>Displacement Source

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Displacement Source

Force Source

The **Force Source** () feature connects a force source between two nodes in the mechanical system.

IDENTIFIER

Enter a **Component name** for the displacement source. The prefix is F.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the force source.

COMPONENT PARAMETERS

Specify the values of **Force**, f_{src} , and **Phase**, ϕ .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**



Theory for Force Source

Context Menus


Lumped Mechanical System>Force Source

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Force Source

External Source

The **External Source** () feature connects an external source between two nodes in the mechanical system.

The equations for the external source can be defined within or outside the physics interface. Typically, an external source can be used to connect a distributed model of a component to the lumped model of a system. The displacement of both the nodes of the distributed component model can be specified as the input on this feature. The reaction forces of both the nodes of the lumped system model can be applied in the distributed component model.

IDENTIFIER

Enter a **Component name** for the external source. The prefix is E.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the external source.

COMPONENT PARAMETERS

Specify the values of **Displacement, port-1**, u_{p10} , and **Displacement, port-2**, u_{p20} .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following in the default plots or evaluations:

- **Force**
- **Displacement**



Theory for External Source

Context Menus


Lumped Mechanical System>External Source

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>External Source

Subsystem Definition

The **Subsystem Definition**  feature is used to define subsystems, which can be inserted as components into the main system using a [Subsystem Instance](#) node. Create the subsystem by adding subnodes to the **Subsystem Definition** node.

NODE CONNECTIONS

Define the number of connecting nodes by adding more nodes or deleting existing nodes and set the Node names at which the subsystem connects to the main system or to other subsystems. The order in which the **Node names** are defined is the order in which they are referenced by a **Subsystem Instance** node.

The components constituting the subsystem should be connected only to the subsystem's nodes and to themselves.

Context Menus


Lumped Mechanical System>Subsystem Definition

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Subsystem Definition

Subsystem Instance

The **Subsystem Instance** () feature represents an instance of a subsystem defined by a **Subsystem Definition** feature.

IDENTIFIER

Enter a **Component name** for the subsystem instance. The prefix is X.

NODE CONNECTIONS

Select the **Name of subsystem link** from the list of defined subsystems in the model and set the **Node names** at which the subsystem instance connects to the main system or to another subsystem.

Context Menus


Lumped Mechanical System>Subsystem Instance

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Lumped Mechanical System>Subsystem Instance

Fixed Node

The **Fixed Node** () feature adds a fixed node to the system. This is the default node in the Lumped Mechanical System interface. This node is used in order to fix one or more ports of a component.



- Disable the **Fixed Node** if none of the nodes of the system is connected to the ground.
- It is also possible to use a single instance of this node to fix the ports of multiple components.

NODE CONNECTIONS

Set the **Node name** for the fixed node.



[Theory for Fixed Node](#)

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Fixed Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Fixed Node

Free Node

The **Free Node** () feature adds a free node to the system. This node is used in order to make one or more ports of a component free.

NODE CONNECTIONS

Set the **Node name** for the free node.



[Theory for Free node](#)

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Free Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Free Node

Displacement Node

The **Displacement Node** () feature adds a displacement node to the system. This node is used in order to prescribe the displacement of one or more ports of a component.

NODE CONNECTIONS

Set the **Node name** for the displacement node.

TERMINAL PARAMETERS

Specify the values of **Displacement**, u_{p10} , and **Phase**, ϕ .



Theory for Displacement Node

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Displacement Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Displacement Node

Velocity Node

The **Velocity Node** () feature adds a velocity node to the system. This node is used in order to prescribe the velocity of one or more ports of a component.

NODE CONNECTIONS

Set the **Node name** for the velocity node.

TERMINAL PARAMETERS

Specify the value of **Velocity**, v_{p1} .

Also select **Displacement in stationary study** — **Free** or **Constrained**.



Theory for Velocity Node

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Velocity Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Velocity Node

Acceleration Node

The **Acceleration Node** () feature adds an acceleration node to the system. This node is used in order to prescribe the acceleration of one or more ports of a component.

NODE CONNECTIONS

Set the **Node name** for the acceleration node.

TERMINAL PARAMETERS

Specify the value of **Acceleration**, a_{p1} .

Also select **Displacement in stationary study** — **Free** or **Constrained**.



Theory for Acceleration Node

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Acceleration Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Acceleration Node

Force Node

The **Force Node** () feature adds a force node to the system. This node is used in order to apply the force at one or more ports of a component.

NODE CONNECTIONS

Set the **Node name** for the force node.

TERMINAL PARAMETERS

Specify the values of **Force**, f_{p10} , and **Phase**, ϕ .



Theory for Force Node

LOCATION IN USER INTERFACE

Context Menus


Lumped Mechanical System>Terminals>Force Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Force Node

Impedance Node

The **Impedance Node** () feature adds an impedance node to the system. This node is used in order to define the impedance of one or more ports of a component.

NODE CONNECTIONS

Set the **Node name** for the impedance node.

TERMINAL PARAMETERS

Specify the value of **Impedance**, Z .



[Theory for Impedance Node](#)

LOCATION IN USER INTERFACE

Context Menus

Lumped Mechanical System>Terminals>Impedance Node

Ribbon

Physics tab with **Lumped Mechanical System** selected:

Global>Terminals>Impedance Node

Theory for the Lumped Mechanical System Interface

This section introduces you to the theory of various components of the Lumped Mechanical System interface.

In this section:

- [Dependent Variables](#)
- [Theory for Two-Port Components](#)
- [Theory for Single-Port Components](#)

Dependent Variables

Every feature of the physics interface adds certain ODE dependent variables in the form of displacement, force, velocity, and others. All the ODE dependent variables are grouped in the following ODE fields:

- Displacements
- Forces
- Velocities
- Other

Theory for Two-Port Components

The equations for all the two port components are described below:

- [Theory for Mass](#)
- [Theory for Spring](#)
- [Theory for Damper](#)
- [Theory for Impedance](#)
- [Theory for Displacement Source](#)
- [Theory for Force Source](#)
- [Theory for External Source](#)

THEORY FOR MASS

A mass component creates a difference in the forces between the two connecting nodes.

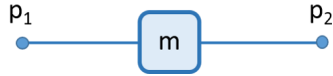


Figure 5-1: A mass component connected between the two nodes p_1 and p_2 .

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned}f_{p1} + f_{p2} &= f \\u_{p2} - u_{p1} &= 0 \\u_{p2} &= u\end{aligned}$$

where f and u are the force and displacement variables in the component. They are related by the following equations, where m is the component mass:

Frequency domain and eigenfrequency studies

$$f = -(i\omega)^2 m u$$

Time dependent studies

$$f = -m \frac{\partial^2 u}{\partial t^2}$$

Stationary studies

$$f = 0$$

An additional gravity force is added in the component force when the gravity contribution is included:

$$f = -mg$$

THEORY FOR SPRING

A spring component creates a difference in the displacements between the two connecting nodes which is proportional to the force.

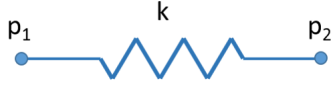


Figure 5-2: A spring component connected between the two nodes $p1$ and $p2$.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned} f_{p1} + f_{p2} &= 0 \\ u_{p2} - u_{p1} &= u \\ f_{p2} &= f \end{aligned}$$

where f and u are the force and displacement variables in the component. They are related by the following equation:

$$f = -k(u - u_0)$$

where k is the spring constant and u_0 is the predeformation.

THEORY FOR DAMPER

A damper component is similar to the spring component and creates a difference in the displacements of the two connecting nodes. The relative velocity is proportional to the force.

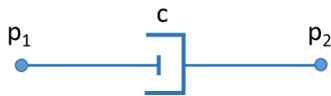


Figure 5-3: A damper component connected between the two nodes $p1$ and $p2$.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned} f_{p1} + f_{p2} &= 0 \\ u_{p2} - u_{p1} &= u \\ f_{p2} &= f \end{aligned}$$

where f and u are the force and displacement variables in the component. They are related by the following equations, where c is the damping coefficient:

Frequency domain and eigenfrequency studies

$$f = -(i\omega)cu$$

Time dependent studies

$$f = -c\frac{\partial u}{\partial t}$$

Stationary studies

$$f = 0$$

THEORY FOR IMPEDANCE

An impedance component creates a difference in the displacements as well as forces between the two connecting nodes.

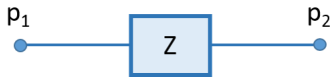


Figure 5-4: An impedance component connected between the two nodes p1 and p2.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned} f_{p1} &= f_1 \\ u_{p1} &= u_1 \\ f_{p2} &= f_2 \\ u_{p2} &= u_2 \end{aligned}$$

where f_1, f_2 and u_1, u_2 are the force and displacement variables in the component. They are related by the following equations:

Frequency domain and eigenfrequency studies

$$\begin{aligned} f_1 &= -(i\omega)(Z_{11}u_1 + Z_{12}u_2) \\ f_2 &= -(i\omega)(Z_{21}u_1 + Z_{22}u_2) \end{aligned}$$

Time dependent and stationary studies

$$\begin{aligned}f_1 &= f_2 \\u_1 &= u_2\end{aligned}$$

Here Z_{11} , Z_{12} , Z_{21} , Z_{22} are the components of the impedance matrix.

THEORY FOR DISPLACEMENT SOURCE

A displacement source creates a difference in the displacements of the two connecting nodes.



Figure 5-5: A displacement source connected between the two nodes p1 and p2.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned}f_{p1} + f_{p2} &= 0 \\u_{p2} - u_{p1} &= u \\f_{p2} &= f\end{aligned}$$

where f and u are the force and displacement variables in the component. The following equation prescribes the component displacement, u :

Frequency domain and eigenfrequency studies

$$u = u_{src} e^{i\phi}$$

Time dependent and stationary studies

$$u = u_{src}$$

Here u_{src} is the prescribed displacement value and ϕ is the phase angle.

THEORY FOR FORCE SOURCE

A force source creates a difference in the forces of the two connecting nodes.

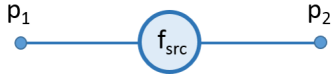


Figure 5-6: A force source connected between the two nodes p1 and p2.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned} f_{p1} + f_{p2} &= f \\ u_{p2} - u_{p1} &= 0 \\ u_{p2} &= u \end{aligned}$$

where f and u are the force and displacement variables in the component. The following equation prescribes the component force, f :

Frequency domain and eigenfrequency studies

$$f = f_{\text{src}} e^{i\phi}$$

Time dependent and stationary studies

$$f = f_{\text{src}}$$

Here, f_{src} is the prescribed force value and ϕ is the phase angle.

THEORY FOR EXTERNAL SOURCE

An external source is similar to the impedance component and creates a difference in the displacements as well as forces of the two connecting nodes.

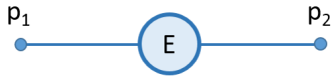


Figure 5-7: An external source connected between the two nodes p1 and p2.

The following equations are used to relate force (f_{p1}) and displacement (u_{p1}) of one node to the force (f_{p2}) and displacement (u_{p2}) of another node:

$$\begin{aligned}
 f_{p1} &= f_1 \\
 u_{p1} &= u_1 \\
 f_{p2} &= f_2 \\
 u_{p2} &= u_2
 \end{aligned}$$

where f_1, f_2 and u_1, u_2 are the force and displacement variables in the component. The following equation prescribes the component displacement (u_1, u_2)

$$\begin{aligned}
 u_1 &= u_{p10} \\
 u_2 &= u_{p20}
 \end{aligned}$$

where u_{p10}, u_{p20} are the displacement values obtained from the component level model.

The component force (f_1, f_2) obtained from the external source node of the system level model can be applied in the component level model as a feedback in order to complete the coupling between the two.

Theory for Single-Port Components

The equations for all the single-port components, which are also known as the terminals, are described below:

- [Theory for Fixed Node](#)
- [Theory for Displacement Node](#)
- [Theory for Velocity Node](#)
- [Theory for Acceleration Node](#)
- [Theory for Free node](#)
- [Theory for Force Node](#)
- [Theory for Impedance Node](#)

THEORY FOR FIXED NODE

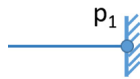


Figure 5-8: A fixed node connected at node p1.

The following equation prescribes the node displacement, u_{p1} :

$$u_{p1} = 0$$

THEORY FOR DISPLACEMENT NODE

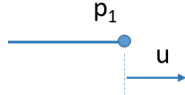


Figure 5-9: A displacement node connected at node $p1$.

The following equation prescribes the node displacement, u_{p1} :

Frequency domain and eigenfrequency studies

$$u_{p1} = u_{p10}e^{i\phi}$$

Time dependent and stationary studies

$$u_{p1} = u_{p10}$$

Here u_{p10} is the prescribed displacement value and ϕ is the phase angle.

THEORY FOR VELOCITY NODE

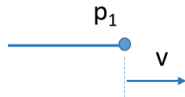


Figure 5-10: A velocity node connected at node $p1$.

The following equation prescribes the node displacement, u_{p1} , given that v_{p1} is the prescribed velocity value:

Frequency domain and eigenfrequency studies

$$(i\omega)u_{p1} = v_{p1}$$

For time dependent studies

$$\frac{\partial u_{p1}}{\partial t} = v_{p1}$$

Stationary studies

If the displacement is set to free:

$$f_{p1} = 0$$

If the displacement is set to constrained:

$$u_{p1} = 0$$

THEORY FOR ACCELERATION NODE

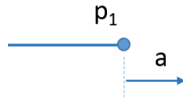


Figure 5-11: An acceleration node connected at node p1.

The following equation prescribes the node displacement, u_{p1} , given that a_{p1} is the prescribed acceleration value:

Frequency domain and eigenfrequency studies

$$(i\omega)^2 u_{p1} = a_{p1}$$

Time dependent studies

$$\frac{\partial^2 u_{p1}}{\partial t^2} = a_{p1}$$

Stationary studies

If the displacement is set to free:

$$f_{p1} = 0$$

If the displacement is set to constrained:

$$u_{p1} = 0$$

THEORY FOR FREE NODE

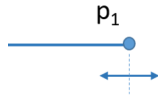


Figure 5-12: A free node connected at node $p1$.

The following equation prescribes the nodal force, f_{p1} :

$$f_{p1} = 0$$

THEORY FOR FORCE NODE

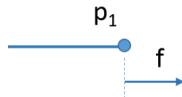


Figure 5-13: A force node connected at node $p1$.

The following equation prescribes the nodal force, f_{p1} :

Frequency domain and eigenfrequency studies

$$f_{p1} = f_{p10}e^{i\phi}$$

Time dependent and stationary studies

$$f_{p1} = f_{p10}$$

Here f_{p10} is the prescribed force value and ϕ is the phase angle.

THEORY FOR IMPEDANCE NODE

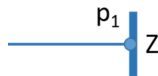


Figure 5-14: An impedance node connected at node $p1$.

The following equation relates the nodal force (f_{p1}) and node displacement (u_{p1}):

Frequency domain and eigenfrequency studies

$$f_{p1} = -(i\omega)Zu_{p1}$$

Time dependent and stationary studies

$$f_{p1} = 0$$

Here Z is the prescribed impedance value.

I n d e x

- A**
 - acceleration node (node) 257
 - activation condition 109
 - Application Libraries window 13
 - applied force and moment (node) 206
 - theory 27, 110
 - attachment (node) 177
 - attachments
 - fixed 21
 - source and destination 21
 - theory 39
 - axial rotation
 - constraints, theory 102
 - locking, theory 105
 - springs and dampers, theory 107
- B**
 - backlash 48
 - backlash (node) 237
 - theory 160
 - ball joint (node) 188
 - theory 35, 88
 - base motion (node) 216
 - bevel gear 58
 - bevel gear (node) 224
 - theory 134
 - boundary nodes 171
- C**
 - cam-follower (node) 217
 - Cardan joint 37, 193
 - theory 37, 94
 - center of joint 23
 - boundary (node) 211
 - edge (node) 211
 - point (node) 211
 - center of rotation
 - attachments 40
 - boundary (node) 181
 - edge (node) 182
 - point (node) 182
 - constraints (node) 197
 - theory 24, 97
 - contact area (node) 210
 - coordinate systems
 - attachments 23
 - ball joints 88
 - cylindrical joints 85
 - hinge joints 82
 - joints 24, 78
 - reduced slot joints 92
 - slot joints 90
 - Coulomb's law 27
 - cylindrical joint (node) 186
 - theory 31, 84
- D**
 - damper (node) 248
 - degrees of freedom 21
 - for different joints 22
 - initial values 29
 - destination attachment 21
 - destination filter (node) 213
 - destination point
 - boundary (node) 212
 - edge (node) 212
 - point (node) 212
 - destination point (node) 216
 - displacement node (node) 255
 - displacement source (node) 250
 - displacement vector 80
 - distance joint (node) 192
 - theory 37, 93
 - documentation 12
 - domain nodes 171
- E**
 - edge nodes 171
 - effective mass 98

- emailing COMSOL 14
- external source (node) 252
- F**
 - fixed attachment 21
 - fixed joint (node) 191
 - theory 37, 93
 - fixed node (node) 254
 - flexible foundation (node) 241
 - fluid-multibody interaction 68, 71
 - force source (node) 251
 - free node (node) 255
 - friction 49
 - friction (node) 207
 - theory 27, 111
 - friction- gear (node)
 - theory 162
 - frictional coefficient 162
 - friction-gear (node) 238
- G**
 - gear axis (node) 228
 - gear elasticity (node) 233
 - theory 152
 - gear geometries 50
 - gear pair (node) 229
 - theory 139
 - gear tooth 51
 - geometric nonlinearity 66
- H**
 - helical gear 56
 - helical gear (node) 223
 - theory 132
 - helical rack 61
 - helical rack (node) 227
 - theory 138
 - hinge joint (node) 185
 - theory 31, 82
 - Hooke's joint 37, 193
 - theory 37, 94
- I**
 - impedance (node) 249
 - impedance node (node) 258
- inclination angle
 - constraints, theory 100
 - locking, theory 105
 - springs and dampers, theory 107
- initial values (node) 175
 - rigid domains 182
- initial values, joints 29
- initially rigid (node) 176
- input shaft orientation (node) 210
- internet resources 11
- J**
 - joint axis (node) 210
 - joint elasticity (node) 195
 - joint forces 28, 118
 - joint moments 28, 118
 - joint types 22
 - joint, definition 21
- K**
 - knowledge base, COMSOL 14
- L**
 - locking (node) 199
 - theory 24, 103
 - lumped mechanical system interface 244
- M**
 - mass (node) 246
 - moving foundation (node) 242
 - MPH-files 13
 - multibody dynamics interface 166
 - multiphysics coupling
 - fluid-multibody interaction 68, 71
- O**
 - output shaft orientation (node) 211
- P**
 - pair nodes 171
 - penalty factor 98
 - penalty method 119
 - planar joint (node) 188
 - theory 34, 87
 - plane stress and strain 167
 - point nodes 171
 - prescribed motion (node) 204
 - theory 26, 108

- prismatic joint (node) 183
 - theory 30, 78
- Q** quaternion representation, of rigid connector 76
 - quaternion vector 80
- R** rack and pinion (node) 232
 - theory 149
- rack axis (node) 228
- rack tooth 52
- radial roller bearing (node)
 - solid rotor 239
- reduced slot joint (node) 190
 - theory 36, 91
- refpnt variable 168
- regularization factor 27, 162
- revolute joint 31, 82
- rigid domain (node) 179
- rotational
 - constraints, theory 100
 - locking, theory 104
 - prescribed motion, theory 109
 - springs and dampers, theory 107
- S** screw joint (node) 187
 - theory 33, 86
- sliding joint 30, 78
- slot joint (node) 189
 - theory 35, 89
- source attachment 21
- source filter (node) 213
- source point
 - boundary (node) 212
 - edge (node) 212
 - point (node) 212
- source point (node) 215
- spring (node) 247
- spring and damper (node) 202
 - theory 25, 106
- spring-damper (node) 214
 - theory 43
- spur gear 53
- spur gear (node) 220
 - theory 130
- spur rack 54, 60
- spur rack (node) 226
 - theory 136
- study steps, geometric nonlinearity and 66
- subsystem definition (node) 253
- subsystem instance (node) 254
- T** technical support, COMSOL 14
- thickness 167
- translational
 - constraints, theory 98
 - locking, theory 103
 - prescribed motion, theory 108
 - springs and dampers, theory 106
- translational joint 30, 78
- transmission error (node) 235
 - theory 159
- U** universal joint
 - theory 37, 94
- universal joint (node) 193
- V** variables
 - refpnt 168
 - velocity node (node) 256
- W** weak constraints 28, 119
- websites, COMSOL 14
- worm and wheel (node) 231
 - theory 147
- worm gear 59
- worm gear (node) 225
 - theory 135

