Contents

Preface xi
  How to read this guide ......................................................... xi

1 ArtiSynth Overview 1
  1.1 System structure .......................................................... 1
    1.1.1 Model components ................................................ 1
    1.1.2 The RootModel ..................................................... 1
    1.1.3 Component path names ............................................ 2
    1.1.4 Model advancement ................................................. 2
    1.1.5 MechModel .......................................................... 3
  1.2 Physics simulation ........................................................ 3
  1.3 Basic packages .......................................................... 5
    1.3.1 maspack ............................................................. 5
    1.3.2 artisynth.core .................................................... 5
    1.3.3 artisynth.demos ................................................... 6
  1.4 Properties ............................................................... 6
    1.4.1 Querying and setting property values ......................... 6
    1.4.2 Property handles and paths ..................................... 6
    1.4.3 Composite and inheritable properties .......................... 7
  1.5 Creating an application model ........................................ 7
    1.5.1 Implementing the build() method ............................. 8
    1.5.2 Making models visible to ArtiSynth .......................... 9
    1.5.3 Loading and running a model ................................... 9

2 Supporting classes 11
  2.1 Vectors and matrices .................................................. 11
  2.2 Rotations and transformations ....................................... 12
  2.3 Points and Vectors ..................................................... 12
  2.4 Spatial vectors and inertias ......................................... 13
  2.5 Meshes ........................................................................ 13
    2.5.1 Mesh creation ....................................................... 14
    2.5.2 Setting normals, colors, and textures ......................... 15
2.5.3 Automatic creation of normals and hard edges ................................................. 17
2.5.4 Vertex and feature coloring ............................................................... 18
2.5.5 Reading and writing mesh files ......................................................... 19
2.5.6 Reading and writing normal and texture information ......................... 21
2.5.7 Constructive solid geometry ............................................................. 21
2.6 Reading source relative files ............................................................... 22
2.7 Reading and caching remote files ......................................................... 24

3 Mechanical Models I ................................................................. 25
3.1 Springs and particles ................................................................. 25
  3.1.1 Axial springs and materials ........................................................ 25
  3.1.2 Example: a simple particle-spring model ....................................... 25
  3.1.3 Dynamic, parametric, and attached components .......................... 27
  3.1.4 Custom axial materials ............................................................... 27
  3.1.5 Damping parameters ................................................................. 28
3.2 Rigid bodies ................................................................. 28
  3.2.1 Frame markers ................................................................. 28
  3.2.2 Example: a simple rigid body-spring model .................................. 29
  3.2.3 Creating rigid bodies ............................................................... 31
  3.2.4 Pose and velocity ................................................................. 31
  3.2.5 Inertia and the surface mesh ...................................................... 32
  3.2.6 Coordinate frames and the center of mass .................................. 33
  3.2.7 Damping parameters ............................................................... 34
  3.2.8 Rendering rigid bodies ............................................................. 35
  3.2.9 Multiple meshes ................................................................. 37
  3.2.10 Example: a composite rigid body ............................................... 38
3.3 Joints and connectors ............................................................. 40
  3.3.1 Joints and coordinate frames ...................................................... 40
  3.3.2 Joint coordinates, constraints, and errors ................................... 41
  3.3.3 Creating Joints ........................................................................ 42
  3.3.4 Working with coordinates .......................................................... 44
  3.3.5 Coordinate limits and locking ...................................................... 45
  3.3.6 Example: a simple hinge joint ...................................................... 46
  3.3.7 Constraint forces ................................................................. 48
  3.3.8 Compliance and regularization .................................................... 49
  3.3.9 Example: an overconstrained linkage ......................................... 51
  3.3.10 Rendering joints ................................................................. 53
3.4 Joint components ............................................................. 54
  3.4.1 Hinge Joint ........................................................................... 54
  3.4.2 Slider Joint ........................................................................... 55
  3.4.3 Cylindrical Joint ................................................................. 56
4.5 Distance Grids and Components .......................................................... 86
4.6 Transforming geometry ................................................................. 88
  4.6.1 Nonlinear transformations ......................................................... 89
  4.6.2 Example: the FemModelDeformer class ................................. 90
  4.6.3 Implementation and behavior .................................................. 92
  4.6.4 Use in model registration ......................................................... 93
4.7 General component arrangements ................................................ 93
  4.7.1 Container components ............................................................ 94
  4.7.2 Example: a net formed from balls and springs ...................... 95
  4.7.3 Adding containers to other models ......................................... 97
4.8 Custom Joints ................................................................................. 97
  4.8.1 Joint constraints .................................................................. 97
  4.8.2 Unilateral constraint engagement ......................................... 99
  4.8.3 Implementing a custom joint ................................................ 101
  4.8.4 Implementing a custom coupling .......................................... 101
  4.8.5 Example: a simple custom joint ......................................... 106
5 Simulation Control ............................................................................. 109
  5.1 Control Panels ........................................................................... 109
    5.1.1 General principles ............................................................. 109
    5.1.2 Example: Creating a simple control panel ....................... 109
  5.2 Custom properties ........................................................................ 111
    5.2.1 Adding properties to a component .................................. 111
    5.2.2 Example: a visibility property ......................................... 112
  5.3 Controllers and monitors ............................................................. 113
    5.3.1 Implementation ................................................................. 113
    5.3.2 Example: A controller to move a point ............................ 114
  5.4 Probes ......................................................................................... 115
    5.4.1 Numeric probe structure .................................................... 116
    5.4.2 Creating probes in code ..................................................... 117
    5.4.3 Example: probes connected to SimpleMuscle ............... 117
    5.4.4 Data file format ................................................................. 118
    5.4.5 Adding probe data in-line ................................................ 120
    5.4.6 Numeric monitor probes ................................................... 121
    5.4.7 Numeric control probes ..................................................... 123
  5.5 Application-Defined Menu Items ............................................... 125
## 6 Finite Element Models

### 6.1 Overview

- 6.1.1 FemModel3d

### 6.1.2 Component Structure

- 6.1.2.1 Nodes
- 6.1.2.2 Elements
- 6.1.2.3 Shell elements
- 6.1.2.4 Meshes

### 6.1.3 Materials

### 6.1.4 Boundary conditions

### 6.2 FEM model creation

- 6.2.1 Factory methods
- 6.2.2 Loading external FEM meshes
- 6.2.3 Generating from surfaces
- 6.2.4 Building elements in code
- 6.2.5 Example: a simple beam model

### 6.3 FEM Geometry

- 6.3.1 Surface meshes
- 6.3.2 Embedding geometry within an FEM
- 6.3.3 Example: a beam with an embedded sphere

### 6.4 Connecting FEM models to other components

- 6.4.1 Connecting nodes to rigid bodies or particles
- 6.4.2 Example: connecting a beam to a block
- 6.4.3 Connecting nodes directly to elements
- 6.4.4 Example: connecting two FEMs together
- 6.4.5 Finding which nodes to attach
- 6.4.6 Example: two bodies connected by an FEM “spring”
- 6.4.7 Nodal-based attachments
- 6.4.8 Example: element vs. nodal-based attachments

### 6.5 FEM markers

- 6.5.1 Example: attaching an FEM beam to a muscle

### 6.6 Frame attachments

- 6.6.1 Example: attaching frames to an FEM beam
- 6.6.2 Adding joints to FEM models
- 6.6.3 Example: two FEM beams connected by a joint

### 6.7 Incompressibility

- 6.7.1 Volume regions and locking
- 6.7.2 Hard incompressibility
- 6.7.3 Soft incompressibility
- 6.7.4 Incompressibility and linear materials
- 6.7.5 Using incompressibility in practice
6.8 Varying and augmenting material behaviors ............................................. 159
  6.8.1 Example: FEM sheet with a stiff spine ............................................... 161
6.9 Muscle activated FEM models ............................................................... 162
  6.9.1 FemMuscleModel ................................................................................ 163
      6.9.1.1 Bundles .................................................................................... 163
      6.9.1.2 Exciters ................................................................................... 163
  6.9.2 Fibre-based muscles .......................................................................... 163
  6.9.3 Material-based muscles ..................................................................... 164
  6.9.4 Example: comparison with two beam examples ................................. 165
6.10 Fields ........................................................................................................ 166
  6.10.1 Grid fields ......................................................................................... 167
  6.10.2 FEM fields ........................................................................................ 169
      6.10.2.1 Nodal fields ............................................................................. 169
      6.10.2.2 Element fields ......................................................................... 170
      6.10.2.3 Sub-element fields ................................................................... 171
  6.10.3 Mesh fields ......................................................................................... 172
      6.10.3.1 Vertex fields ............................................................................ 173
      6.10.3.2 Face fields ............................................................................... 174
  6.10.4 Fields for VectorNd, MatrixNd and Vector3d ..................................... 175
  6.10.5 Binding material properties ............................................................... 176
  6.10.6 Example: FEM with variable stiffness ............................................ 177
  6.10.7 Example: specifying FEM muscle directions .................................... 178
6.11 Rendering and Visualizations ................................................................. 180
  6.11.1 Example: stress and strain plotting ............................................... 182
7 Contact and Collision ................................................................................. 185
  7.1 Enabling collisions .................................................................................. 185
      7.1.1 Collisions between specific bodies ................................................. 186
      7.1.2 Default collisions between groups .................................................. 187
      7.1.3 Example: collision with a plane ..................................................... 188
      7.1.4 Collisions for FEM models ............................................................ 189
      7.1.5 Example: FEM models and rigid bodies .................................... 190
  7.2 Collision behaviors and collidability ...................................................... 190
      7.2.1 Collision behaviors ....................................................................... 190
      7.2.2 Collidability .................................................................................. 193
  7.3 Collision meshes and compound collidables ........................................ 193
      7.3.1 Example: redefining a rigid body collision mesh ......................... 194
      7.3.2 Compound collidables and self-collision ....................................... 195
      7.3.3 Example: FEM model with self-collision ..................................... 196
      7.3.4 Collidable bodies .......................................................................... 198
      7.3.5 Nested MechModels .................................................................... 199
7.4 Implementation ...................................................... 200
  7.4.1 Contact methods .............................................. 200
    7.4.1.1 Contour region ......................................... 200
    7.4.1.2 Vertex penetration ..................................... 201
    7.4.1.3 Setting the contact method .................................. 201
  7.4.2 Collider types .............................................. 203
    7.4.2.1 Collision meshes and signed distance grids ................. 203
7.5 Contact rendering ............................................. 204
  7.5.1 Example: rendering normals and contours ......................... 207
  7.5.2 Example: rendering a color map .................................. 209
  7.5.3 Example: rendering contact pressures .......................... 212
7.6 Overconstrained contact ......................................... 215
  7.6.1 Constraint reduction ........................................ 216
7.7 Contact regularization ........................................... 216
  7.7.1 Compliant contact ......................................... 216
  7.7.2 Contact force behaviors .................................... 217
    7.7.2.1 Computing forces based on pressure ....................... 219
  7.7.3 Elastic foundation contact .................................... 220
  7.7.4 Example: elastic foundation contact of a ball in a bowl ......... 221
  7.7.5 Example: binding EFC properties to fields ..................... 222
7.8 Monitoring collisions ............................................ 225
  7.8.1 Collision responses ...................................... 225
  7.8.2 Available information ................................... 225
  7.8.3 Example: monitoring contact forces ........................ 227
7.9 Tips and limitations ............................................. 229
  7.9.1 Contact jitter .................................. 229
  7.9.2 Passing through objects .................................. 229
  7.9.3 Stray vertices ........................................ 230
  7.9.4 Coulomb friction and stability .................................. 230
8 Muscle Wrapping and Via Points .................................... 231
  8.1 Via Points .............................................. 232
    8.1.1 Example: a muscle with via points ......................... 233
  8.2 Obstacle Wrapping ........................................... 235
    8.2.1 Example: wrapping around a cylinder ....................... 236
  8.3 General Surfaces and Distance Grids ................................ 239
    8.3.1 Example: wrapping around a bone ........................ 240
  8.4 Initializing the Wrap Path ..................................... 242
    8.4.1 Example: wrapping around a torus ........................ 242
  8.5 Alternate Wrapping Surfaces .................................... 244
    8.5.1 Example: wrapping for a finger joint ....................... 244
  8.6 Tuning the Wrapping Behavior ................................... 246
Preface

This guide describes how to create mechanical and biomechanical models in ArtiSynth using its Java API. Detailed information on how to use the ArtiSynth GUI for model visualization, navigation and simulation control is given in the ArtiSynth User Interface Guide. It is also possible to interface ArtiSynth with, or run it under, MATLAB. For information on this, see the guide Interfacing ArtiSynth to MATLAB.

Information on how to install and configure ArtiSynth is given in the installation guides for Windows, MacOS, and Linux.

It is assumed that the reader is familiar with basic Java programming, including variable assignment, control flow, exceptions, functions and methods, object construction, inheritance, and method overloading. Some familiarity with the basic I/O classes defined in java.io.*, including input and output streams and the specification of file paths using File, as well as the collection classes ArrayList and LinkedList defined in java.util.*, is also assumed.

How to read this guide

Section 1 offers a general overview of ArtiSynth’s software design, and briefly describes the algorithms used for physical simulation (Section 1.2). The latter section may be skipped on first reading. A more comprehensive overview paper is available online.

The remainder of the manual gives details instructions on how to build various types of mechanical and biomechanical models. Sections 3 and 4 give detailed information about building general mechanical models, involving particles, springs, rigid bodies, joints, constraints, and contact. Section 5 describes how to add control panels, controllers, and input and output data streams to a simulation. Section 6 describes how to incorporate finite element models. The required mathematics is reviewed in Section A.

If time permits, the reader will profit from a top-to-bottom read. However, this may not always be necessary. Many of the sections contain detailed examples, all of which are available in the package artisynth.demos.tutorial and which may be run from ArtiSynth using Models > All demos > tutorials. More experienced readers may wish to find an appropriate example and then work backwards into the text and preceding sections for any needed explanatory detail.
Chapter 1

ArtiSynth Overview

ArtiSynth is an open-source, Java-based system for creating and simulating mechanical and biomechanical models, with specific capabilities for the combined simulation of rigid and deformable bodies, together with contact and constraints. It is presently directed at application domains in biomechanics, medicine, physiology, and dentistry, but it can also be applied to other areas such as traditional mechanical simulation, ergonomic design, and graphical and visual effects.

1.1 System structure

An ArtiSynth model is composed of a hierarchy of models and model components which are implemented by various Java classes. These may include sub-models (including finite element models), particles, rigid bodies, springs, connectors, and constraints. The component hierarchy may be in turn connected to various agent components, such as control panels, controllers and monitors, and input and output data streams (i.e., probes), which have the ability to control and record the simulation as it advances in time. Agents are presented in more detail in Section 5.

The models and agents are collected together within a top-level component known as a root model. Simulation proceeds under the control of a scheduler, which advances the models through time using a physics simulator. A rich graphical user interface (GUI) allows users to view and edit the model hierarchy, modify component properties, and edit and temporally arrange the input and output probes using a timeline display.

1.1.1 Model components

Every ArtiSynth component is an instance of ModelComponent. When connected to the hierarchy, it is assigned a unique number relative to its parent; the parent and number can be obtained using the methods getParent() and getNumber(), respectively. Components may also be assigned a name (using setName()) which is then returned using getName().

A sub-interface of ModelComponent includes CompositeComponent, which contains child components. A ComponentList is a CompositeComponent which simply contains a list of other components (such as particles, rigid bodies, sub-models, etc.).

Components which contain state information (such as position and velocity) should extend HasState, which provides the methods getState() and setState() for saving and restoring state.

A Model is a sub-interface of CompositeComponent and HasState that contains the notion of advancing through time and which implements this with the methods initialize(t0) and advance(t0, t1, flags), as discussed further in Section 1.1.4. The most common instance of Model used in ArtiSynth is MechModel (Section 1.1.5), which is the top-level container for a mechanical or biomechanical model.

1.1.2 The RootModel

The top-level component in the hierarchy is the root model, which is a subclass of RootModel and which contains a list of models along with lists of agents used to control and interact with these models. The component lists in RootModel include:
Each agent may be associated with a specific top-level-level model.

### 1.1.3 Component path names

The names and/or numbers of a component and its ancestors can be used to form a component path name. This path has a construction analogous to Unix file path names, with the '/' character acting as a separator. Absolute paths start with '/', which indicates the root model. Relative paths omit the leading '/' and can begin lower down in the hierarchy. A typical path name might be

```
/models/JawHyoidModel/axialSprings/1ad
```

For nameless components in the path, their numbers can be used instead. Numbers can also be used for components that have names. Hence the path above could also be represented using only numbers, as in

```
/0/0/1/5
```

although this would most likely appear only in machine-generated output.

### 1.1.4 Model advancement

ArtiSynth simulation proceeds by advancing all of the root model’s top-level models through a sequence of time steps. Every time step is achieved by calling each model’s `advance()` method:

```java
public StepAdjustment advance (double t0, double t1) {
  ...
  perform simulation ...
  return; // optional step adjustment
}
```

This method advances the model from time $t_0$ to time $t_1$, performing whatever physical simulation is required (see Section 1.2). The method may optionally return a `StepAdjustment` indicating that the step size $(t_1 - t_0)$ was too large and that the advance should be redone with a smaller step size.

The root model has its own `advance()`, which in turn calls the advance method for all of the top-level models, in sequence. The advance of each model is surrounded by the application of whatever agents are associated with that model. This is done by calling the agent’s `apply()` method:

```java
model.preadvance (t0, t1);
for (each input probe p) {
  p.apply (t1);
}
for (each controller c) {
  c.apply (t0, t1);
}
model.advance (t0, t1);
for (each monitor m) {
  m.apply (t0, t1);
}
for (each output probe p) {
  p.apply (t1);
}
```

Agents not associated with a specific model are applied before (or after) the advance of all other models.

More precise details about model advancement are given in the [ArtiSynth Reference Manual](#).
1.1.5 MechModel

Most ArtiSynth applications contain a single top-level model which is an instance of MechModel. This is a CompositeComponent that may (recursively) contain an arbitrary number of mechanical components, including finite element models, other MechModels, particles, rigid bodies, constraints, attachments, and various force effectors. The MechModel advance() method invokes a physics simulator that advances these components forward in time (Section 1.2).

For convenience each MechModel contains a number of predefined containers for different component types, including:

<table>
<thead>
<tr>
<th>Component Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>particles</td>
<td>3 DOF particles</td>
</tr>
<tr>
<td>points</td>
<td>other 3 DOF points</td>
</tr>
<tr>
<td>rigidBodies</td>
<td>6 DOF rigid bodies</td>
</tr>
<tr>
<td>frames</td>
<td>other 6 DOF frames</td>
</tr>
<tr>
<td>axialSprings</td>
<td>point-to-point springs</td>
</tr>
<tr>
<td>connectors</td>
<td>joint-type connectors between bodies</td>
</tr>
<tr>
<td>constrainers</td>
<td>general constraints</td>
</tr>
<tr>
<td>forceEffectors</td>
<td>general force-effectors</td>
</tr>
<tr>
<td>attachments</td>
<td>attachments between dynamic components</td>
</tr>
<tr>
<td>renderables</td>
<td>renderable components (for visualization only)</td>
</tr>
</tbody>
</table>

Each of these is a child component of MechModel and is implemented as a ComponentList. Special methods are provided for adding and removing items from them. However, applications are not required to use these containers, and may instead create any component containment structure that is appropriate. If not used, the containers will simply remain empty.

1.2 Physics simulation

Only a brief summary of ArtiSynth physics simulation is described here. Full details are given in [9] and in the related overview paper.

For purposes of physics simulation, the components of a MechModel are grouped as follows:

Dynamic components
- Components, such as a particles and rigid bodies, that contain position and velocity state, as well as mass. All dynamic components are instances of the Java interface DynamicComponent.

Force effectors
- Components, such as springs or finite elements, that exert forces between dynamic components. All force effectors are instances of the Java interface ForceEffector.

Constrainers
- Components that enforce constraints between dynamic components. All constrainers are instances of the Java interface Constrainer.

Attachments
- Attachments between dynamic components. While technically these are constraints, they are implemented using a different approach. All attachment components are instances of DynamicAttachment.

The positions, velocities, and forces associated with all the dynamic components are denoted by the composite vectors \( \mathbf{q}, \mathbf{u}, \) and \( \mathbf{f} \). In addition, the composite mass matrix is given by \( \mathbf{M} \). Newton’s second law then gives

\[
\frac{d\mathbf{u}}{dt} = \mathbf{f} + \mathbf{Mu},
\]  

where the \( \mathbf{Mu} \) accounts for various “fictitious” forces.

Each integration step involves solving for the velocities \( \mathbf{u}^{k+1} \) at time step \( k + 1 \) given the velocities and forces at step \( k \). One way to do this is to solve the expression

\[
\mathbf{Mu}^{k+1} = \mathbf{Mu}^k + h\mathbf{f}
\]  

(1.2)
for \( \mathbf{u}^{k+1} \), where \( h \) is the step size and \( \mathbf{f} \equiv f - \mathbf{M} \mathbf{u} \). Given the updated velocities \( \mathbf{u}^{k+1} \), one can determine \( \mathbf{q}^{k+1} \) from
\[
\mathbf{q}^{k+1} = \mathbf{Q} \mathbf{u}^{k+1},
\]
(1.3)
where \( \mathbf{Q} \) accounts for situations (like rigid bodies) where \( \dot{\mathbf{q}} \neq \mathbf{u} \), and then solve for the updated positions using
\[
\mathbf{q}^{k+1} = \mathbf{q}^k + h \dot{\mathbf{q}}^{k+1}.
\]
(1.4)
(1.2) and (1.4) together comprise a simple symplectic Euler integrator.

In addition to forces, bilateral and unilateral constraints give rise to locally linear constraints on \( \mathbf{u} \) of the form
\[
\mathbf{G}(\mathbf{q}) \mathbf{u} = 0, \quad \mathbf{N}(\mathbf{q}) \mathbf{u} \geq 0.
\]
(1.5)
Bilateral constraints may include rigid body joints, FEM incompressibility, and point-surface constraints, while unilateral constraints include contact and joint limits. Constraints give rise to constraint forces (in the directions \( \mathbf{G}(\mathbf{q})^T \) and \( \mathbf{N}(\mathbf{q})^T \)) which supplement the forces of (1.1) in order to enforce the constraint conditions. In addition, for unilateral constraints, we have a complementarity condition in which \( \mathbf{N} \mathbf{u} > 0 \) implies no constraint force, and a constraint force implies \( \mathbf{N} \mathbf{u} = 0 \). Any given constraint usually involves only a few dynamic components and so \( \mathbf{G} \) and \( \mathbf{N} \) are generally sparse.

Adding constraints to the velocity solve (1.2) leads to a mixed linear complementarity problem (MLCP) of the form
\[
\mathbf{M}^k - \mathbf{G}^T - \mathbf{N}^T \begin{pmatrix} \frac{\lambda}{\theta} \\ \delta \end{pmatrix} + \begin{pmatrix} -\mathbf{f}^k \\ -\mathbf{g} \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \end{pmatrix},
\]
(1.6)
where \( \mathbf{w} \) is a slack variable, \( \lambda \) and \( \delta \) give the force constraint impulses over the time step, and \( \mathbf{g} \) and \( \mathbf{n} \) are derivative terms defined by
\[
\mathbf{g} \equiv -h \mathbf{G} \mathbf{u}^k, \quad \mathbf{n} \equiv -h \mathbf{N} \mathbf{u}^k,
\]
(1.7)
to account for time variations in \( \mathbf{G} \) and \( \mathbf{N} \). In addition, \( \mathbf{M}^k \) and \( \mathbf{f}^k \) are \( \mathbf{M} \) and \( \mathbf{f} \) augmented with stiffness and damping terms to accommodate implicit integration, which is often required for problems involving deformable bodies. The actual constraint forces \( \lambda \) and \( \theta \) can be determined by dividing the impulses by the time step \( h \):
\[
\lambda = \frac{\lambda}{h}, \quad \theta = \frac{\delta}{h}.
\]
(1.8)
We note here that ArtiSynth uses a full coordinate formulation, in which the position of each dynamic body is solved using full, or unconstrained, coordinates, with constraint relationships acting to restrict these coordinates. In contrast, some other simulation systems, including OpenSim [5], use reduced coordinates, in which the system dynamics are formulated using a smaller set of coordinates (such as joint angles) that implicitly take the system’s constraints into account. Each methodology has its own advantages. Reduced formulations yield systems with fewer degrees of freedom and no constraint errors. On the other hand, full coordinates make it easier to combine and connect a wide range of components, including rigid bodies and FEM models.

Attachments between components can be implemented by constraining the velocities of the attached components using special constraints of the form
\[
\mathbf{u}_j = -\mathbf{G}_{ja} \mathbf{u}_a
\]
(1.9)
where \( \mathbf{u}_j \) and \( \mathbf{u}_a \) denote the velocities of the attached and non-attached components. The constraint matrix \( \mathbf{G}_{ja} \) is sparse, with a non-zero block entry for each master component to which the attached component is connected. The simplest case involves attaching a point \( j \) to another point \( k \), with the simple velocity relationship
\[
\mathbf{u}_j = \mathbf{u}_k
\]
(1.10)
That means that \( \mathbf{G}_{ja} \) has a single entry of \(-\mathbf{I}\) (where \( \mathbf{I} \) is the 3 \times 3 identity matrix) in the \( k \)-th block column. Another common case involves connecting a point \( j \) to a rigid frame \( k \). The velocity relationship for this is
\[
\mathbf{u}_j = \mathbf{u}_k - \mathbf{l}_j \times \omega_k
\]
(1.11)
where \( \mathbf{u}_k \) and \( \omega_k \) are the translational and rotational velocity of the frame and \( l_j \) is the location of the point relative to the frame’s origin (as seen in world coordinates). The corresponding \( \mathbf{G}_{ja} \) contains a single 3 \times 6 block entry of the form
\[
\begin{pmatrix} \mathbf{I} & [l_j] \end{pmatrix}
\]
(1.12)
in the \( k^{th} \) block column, where

\[
\begin{bmatrix}
0 & -l_z & l_y \\
-l_z & 0 & -l_x \\
l_y & l_x & 0
\end{bmatrix}
\] (1.13)

is a skew-symmetric cross product matrix. The attachment constraints \( G_{j\alpha} \) could be added directly to (1.6), but their special form allows us to explicitly solve for \( \mathbf{u}_j \), and hence reduce the size of (1.6), by factoring out the attached velocities before solution.

The MLCP (1.6) corresponds to a single step integrator. However, higher order integrators, such as Newmark methods, usually give rise to MLCPs with an equivalent form. Most ArtiSynth integrators use some variation of (1.6) to determine the system velocity at each time step.

To set up (1.6), the MechModel component hierarchy is traversed and the methods of the different component types are queried for the required values. Dynamic components (type DynamicComponent) provide \( \mathbf{q}, \mathbf{u}, \) and \( \mathbf{M} \); force effectors (ForceEffector) determine \( \hat{\mathbf{f}} \) and the stiffness/damping augmentation used to produce \( \hat{\mathbf{M}} \); constrainers (Constrainer) supply \( \mathbf{G}, \mathbf{N}, \mathbf{g} \) and \( \mathbf{n} \), and attachments (DynamicAttachment) provide the information needed to factor out attached velocities.

### 1.3 Basic packages

The core code of the ArtiSynth project is divided into three main packages, each with a number of sub-packages.

#### 1.3.1 maspack

The packages under maspack contain general computational utilities that are independent of ArtiSynth and could be used in a variety of other contexts. The main packages are:

- maspack.util // general utilities
- maspack.matrix // matrix and linear algebra
- maspack.graph // graph algorithms
- maspack.fileutil // remote file access
- maspack.properties // property implementation
- maspack.spatialmotion // 3D spatial motion and dynamics
- maspack.solvers // LCP solvers and linear solver interfaces
- maspack.render // viewer and rendering classes
- maspack.geometry // 3D geometry and meshes
- maspackcollision // collision detection
- maspack.widgets // Java swing widgets for maspack data types
- maspack.apps // stand-alone programs based only on maspack

#### 1.3.2 artisynth.core

The packages under artisynth.core contain the core code for ArtiSynth model components and its GUI infrastructure.

- artisynth.core.util // general ArtiSynth utilities
- artisynth.core.modelbase // base classes for model components
- artisynth.core.materials // materials for springs and finite elements
- artisynth.core.mechmodels // basic mechanical models
- artisynth.core.femmodels // finite element models
- artisynth.core.probes // input and output probes
- artisynth.core.workspace // RootModel and associated components
- artisynth.core.driver // start ArtiSynth and drive the simulation
- artisynth.core.gui // graphical interface
- artisynth.core.inverse // inverse tracking controller
- artisynth.core.moviemaker // used for making movies
- artisynth.core.renderables // components that are strictly visual
- artisynth.core.opensim // OpenSim model parser (under development)
- artisynth.core.mfreemodels // mesh free models (experimental, not supported)
1.3.3 artisynth.demos

These packages contain demonstration models that illustrate ArtiSynth’s modeling capabilities:

<table>
<thead>
<tr>
<th>Package</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>artisynth.demos.mech</td>
<td>mechanical model demos</td>
</tr>
<tr>
<td>artisynth.demos.fem</td>
<td>demos involving finite elements</td>
</tr>
<tr>
<td>artisynth.demos.inverse</td>
<td>demos involving inverse control</td>
</tr>
<tr>
<td>artisynth.demos.tutorial</td>
<td>demos in this manual</td>
</tr>
</tbody>
</table>

1.4 Properties

ArtiSynth components expose properties, which provide a uniform interface for accessing their internal parameters and state. Properties vary from component to component; those for RigidBody include position, orientation, mass, and density, while those for AxialSpring include restLength and material. Properties are particularly useful for automatically creating control panels and probes, as described in Section 5. They are also used for automating component serialization.

Properties are described only briefly in this section; more detailed descriptions are available in the Maspack Reference Manual and the overview paper.

The set of properties defined for a component is fixed for that component’s class; while property values may vary between component instances, their definitions are class-specific. Properties are exported by a class through code contained in the class definition, as described in Section 5.2.

1.4.1 Querying and setting property values

Each property has a unique name that can be used to access its value interactively in the GUI. This can be done either by using a custom control panel (Section 5.1) or by selecting the component and choosing Edit properties ... from the right-click context menu).

Properties can also be accessed in code using their set/get accessor methods. Unless otherwise specified, the names for these are formed by simply prepending set or get to the property’s name. More specifically, a property with the name foo and a value type of Bar will usually have accessor signatures of

```java
Bar getFoo()
void setFoo (Bar value)
```

1.4.2 Property handles and paths

A property’s name can also be used to obtain a property handle through which its value may be queried or set generically. Property handles are implemented by the class Property and are returned by the component’s getProperty() method. getProperty() takes a property’s name and returns the corresponding handle. For example, components of type Muscle have a property excitation, for which a handle may be obtained using a code fragment such as

```java
Muscle muscle;
...
Property prop = muscle.getProperty ("excitation");
```

Property handles can also be obtained for subcomponents, using a property path that consists of a path to the subcomponent followed by a colon ':' and the property name. For example, to obtain the excitation property for a subcomponent located by axialSprings/lad relative to a MechModel, one could use a call of the form

```java
MechModel mech;
...
Property prop = mech.getProperty ("axialSprings/lad:excitation");
```
1.4.3 Composite and inheritable properties

Composite properties are possible, in which a property value is a composite object that in turn has subproperties. A good example of this is the RenderProps class, which is associated with the property renderProps for renderable objects and which itself can have a number of subproperties such as visible, faceStyle, faceColor, lineStyle, lineColor, etc.

Properties can be declared to be inheritable, so that their values can be inherited from the same properties hosted by ancestor components further up the component hierarchy. Inheritable properties require a more elaborate declaration and are associated with a mode which may be either Explicit or Inherited. If a property’s mode is inherited, then its value is obtained from the closest ancestor exposing the same property whose mode is explicit. In Figure (1.1), the property stiffness is explicitly set in components A, C, and E, and inherited in B and D (which inherit from A) and F (which inherits from C).

1.5 Creating an application model

ArtiSynth applications are created by writing and compiling an application model that is a subclass of RootModel. This application-specific root model is then loaded and run by the ArtiSynth program.

The code for the application model should:

- Declare a no-args constructor
- Override the RootModel build() method to construct the application.

ArtiSynth can load a model either using the build method or by reading it from a file:

**Build method**

ArtiSynth creates an instance of the model using the no-args constructor, assigns it a name (which is either user-specified or the simple name of the class), and then calls the build() method to perform the actual construction.

**Reading from a file**

ArtiSynth creates an instance of the model using the no-args constructor, and then the model is named and constructed by reading the file.

The no-args constructor should perform whatever initialization is required in both cases, while the build() method takes the place of the file specification. Unless a model is originally created using a file specification (which is very tedious), the first time creation of a model will almost always entail using the build() method.

The general template for application model code looks like this:
Here, the model itself is called `MyModel`, and is defined in the (hypothetical) package `artisynth.models.experimental` (placing models in the super package `artisynth.models` is common practice but not necessary).

Note: The `build()` method was only introduced in ArtiSynth 3.1. Prior to that, application models were constructed using a constructor taking a `String` argument supplying the name of the model. This method of model construction still works but is deprecated.

### 1.5.1 Implementing the `build()` method

As mentioned above, the `build()` method is responsible for actual model construction. Many applications are built using a single top-level `MechModel`. Build methods for these may look like the following:

```java
public void build (String[] args) {
    MechModel mech = new MechModel("mech");
    addModel (mech);

    ... create and add components to the mech model ...
    ... create and add any needed agents to the root model ...
}
```

First, a `MechModel` is created (with the name "mech" in this example, although any name, or no name, may be given) and added to the list of models in the root model using the `addModel()` method. Subsequent code then creates and adds the components required by the `MechModel`, as described in Sections 3, 4 and 6. The `build()` method also creates and adds to the root model any agents required by the application (controllers, probes, etc.), as described in Section 5.

When constructing a model, there is no fixed order in which components need to be added. For instance, in the above example, `addModel(mech)` could be called near the end of the `build()` method rather than at the beginning. The only restriction is that when a component is added to the hierarchy, all other components that it refers to should already have been added to the hierarchy. For instance, an axial spring (Section 3.1) refers to two points. When it is added to the hierarchy, those two points should already be present in the hierarchy.

The `build()` method supplies a `String` array as an argument, which can be used to transmit application arguments in a manner analogous to the `args` argument passed to static `main()` methods. Build arguments can be specified when a model is loaded directly from a class using `Models > Load from class ...`, or when the `startup model` is set to automatically load a model when ArtiSynth is first started (Settings > Startup model). Details are given in the “Loading, Simulating and Saving Models” section of the User Interface Guide.

Build arguments can also be listed directly on the ArtiSynth command line when specifying a model to load using the `-model <classname>` option. This is done by enclosing the desired arguments within square brackets `[]` immediately following the `-model` option. So, for example,

```shell
> artisynth -model projects.MyModel [ -size 50 ]
```

would cause the strings "-size" and "50" to be passed to the `build()` method of `MyModel`.
1.5.2 Making models visible to ArtiSynth

In order to load an application model into ArtiSynth, the classes associated with its implementation must be made visible to ArtiSynth. This usually involves adding the top-level class folder associated with the application code to the classpath used by ArtiSynth.

The demonstration models referred to in this guide belong to the package `artisynth.demos.tutorial` and are already visible to ArtiSynth.

In most current ArtiSynth projects, classes are stored in a folder tree separate from the source code, with the top-level class folder named `classes`, located one level below the project root folder. A typical top-level class folder might be stored in a location like this:

```
/home/joeuser/artisynthProjects/classes
```

In the example shown in Section 1.5, the model was created in the package `artisynth.models.experimental`. Since Java classes are arranged in a folder structure that mirrors package names, with respect to the sample project folder shown above, the model class would be located in

```
/home/joeuser/artisynthProjects/classes/artisynth/models/experimental
```

At present there are three ways to make top-level class folders known to ArtiSynth:

Add projects to your Eclipse launch configuration

If you are using the Eclipse IDE, then you can add the project in which are developing your model code to the launch configuration that you use to run ArtiSynth. Other IDEs will presumably provide similar functionality.

Add the folders to the external classpath

You can explicitly add the class folders to ArtiSynth’s external classpath. The easiest way to do this is to select “Settings > External classpath ...” from the Settings menu, which will open an external classpath editor which lists all the classpath entries in a large panel on the left. (When ArtiSynth is first installed, the external classpath has no entries, and so this panel will be blank.) Class folders can then be added via the “Add class folder” button, and the classpath is saved using the Save button.

Add the folders to your CLASSPATH environment variable

If you are running ArtiSynth from the command line, using the `artisynth` command (or `artisynth.bat` on Windows), then you can define a CLASSPATH environment variable in your environment and add the needed folders to this.

All of these methods are described in more detail in the “Installing External Models and Packages” section of the ArtiSynth Installation Guide (available for Linux, Windows, and MacOS).

1.5.3 Loading and running a model

If a model’s classes are visible to ArtiSynth, then it may be loaded into ArtiSynth in several ways:

Loading from the Model menu

If the root model is contained in a package located under `artisynth.demos` or `artisynth.models`, then it will appear in the default model menu (Models in the main menu bar) under the submenu All demos or All models.

Loading by class path

A model may also be loaded by choosing “Load from class ...” from the Models menu and specifying its package name and then choosing its root model class. It is also possible to use the `-model <classname>` command line argument to have a model loaded directly into ArtiSynth when it starts up.
Loading from a file

If a model has been saved to a .art file, it may be loaded from that file by choosing File > Load model ....

These methods are described in detail in the section “Loading and Simulating Models” of the ArtiSynth User Interface Guide.

The demonstration models referred to in this guide should already be present in the model menu and may be loaded from the submenu Models > All demos > tutorial.

Once a model is loaded, it can be simulated, or run. Simulation of the model can then be started, paused, single-stepped, or reset using the play controls (Figure 1.2) located at the upper right of the ArtiSynth window frame. Starting and stopping a simulation is done by clicking play/pause, while reset resets the simulation to time 0. The single-step button advances the simulation by one time step. The stop-all button will also stop the simulation, along with any Jython commands or scripts that are running.

![Figure 1.2: The ArtiSynth play controls. From left to right: step size control, current simulation time, and the reset, skip-back, play/pause, single-step, skip-forward and stop-all buttons.](image)

Comprehensive information on exploring and interacting with models is given in the ArtiSynth User Interface Guide.
Chapter 2

Supporting classes

ArtiSynth uses a large number of supporting classes, mostly defined in the super package maspack, for handling mathematical and geometric quantities. Those that are referred to in this manual are summarized in this section.

2.1 Vectors and matrices

Among the most basic classes are those used to implement vectors and matrices, defined in maspack.matrix. All vector classes implement the interface Vector and all matrix classes implement Matrix, which provide a number of standard methods for setting and accessing values and reading and writing from I/O streams.

General sized vectors and matrices are implemented by VectorNd and MatrixNd. These provide all the usual methods for linear algebra operations such as addition, scaling, and multiplication:

```
VectorNd v1 = new VectorNd(5); // create a 5 element vector
VectorNd v2 = new VectorNd(5);
VectorNd vr = new VectorNd(5);
MatrixNd M = new MatrixNd(5, 5); // create a 5 x 5 matrix

M.setIdentity(); // M = I
M.scale(4); // M = 4*M
v1.set(new double[5] {1, 2, 3, 4, 5}); // set values
v2.set(new double[5] {0, 1, 2, 3, 4});
v1.add(v2); // v1 += v2
M.mul(vr, v1); // vr = M*v1

System.out.println("result= " + vr.toString("%8.3f"));
```

As illustrated in the above example, vectors and matrices both provide a toString() method that allows their elements to be formatted using a C-printf style format string. This is useful for providing concise and uniformly formatted output, particularly for diagnostics. The output from the above example is

```
result= 4.000 12.000 12.000 24.000 20.000
```

Detailed specifications for the format string are provided in the documentation for NumberFormat.set(String). If either no format string, or the string "%g", is specified, toString() formats all numbers using the full-precision output provided by Double.toString(value).

For computational efficiency, a number of fixed-size vectors and matrices are also provided. The most commonly used are those defined for three dimensions, including Vector3d and Matrix3d:

```
Vector3d v1 = new Vector3d(1, 2, 3);
Vector3d v2 = new Vector3d(3, 4, 5);
Vector3d vr = new Vector3d();
Matrix3d M = new Matrix3d();
```
2.2 Rotations and transformations

maspack.matrix contains a number classes that implement rotation matrices, rigid transforms, and affine transforms.

Rotations (Section A.1) are commonly described using a RotationMatrix3d, which implements a rotation matrix and contains numerous methods for setting rotation values and transforming other quantities. Some of the more commonly used methods are:

```
RotationMatrix3d();  // create and set to the identity
RotationMatrix3d(u, angle); // create and set using an axis-angle
```

```
setAxisAngle (u, ang);  // set using an axis-angle
setRpy (roll, pitch, yaw); // set using roll-pitch-yaw angles
setEuler (phi, theta, psi); // set using Euler angles
invert ();  // invert this rotation
mul (R);  // post multiply this rotation by R
mul (R1, R2);  // set this rotation to R1*R2
mul (vr, vl);  // vr = R*v1, where R is this rotation
```

Rotations can also be described by AxisAngle, which characterizes a rotation as a single rotation about a specific axis.

Rigid transforms (Section A.2) are used by ArtiSynth to describe a rigid body’s pose, as well as its relative position and orientation with respect to other bodies and coordinate frames. They are implemented by RigidTransform3d, which exposes its rotational and translational components directly through the fields R (a RotationMatrix3d) and p (a Vector3d). Rotational and translational values can be set and accessed directly through these fields. In addition, RigidTransform3d provides numerous methods, some of the more commonly used of which include:

```
RigidTransform3d();  // create and set to the identity
RigidTransform3d(x, y, z); // create and set translation to x, y, z

// create and set translation to x, y, z and rotation to roll-pitch-yaw
RigidTransform3d(x, y, z, roll, pitch, yaw);
```

```
invert ();  // invert this transform
mul (T);  // post multiply this transform by T
mul (T1, T2);  // set this transform to T1*T2
mulLeftInverse (T1, T2);  // set this transform to inv(T1)*T2
```

Affine transforms (Section A.3) are used by ArtiSynth to effect scaling and shearing transformations on components. They are implemented by AffineTransform3d.

Rigid transformations are actually a specialized form of affine transformation in which the basic transform matrix equals a rotation. RigidTransform3d and AffineTransform3d hence both derive from the same base class AffineTransform3dBase.

2.3 Points and Vectors

The rotations and transforms described above can be used to transform both vectors and points in space.

Vectors are most commonly implemented using Vector3d, while points can be implemented using the subclass Point3d. The only difference between Vector3d and Point3d is that the former ignores the translational component of rigid and
affine transforms; i.e., as described in Sections A.2 and A.3, a vector $v$ has an implied homogeneous representation of

$$v^* \equiv \begin{pmatrix} v \\ 0 \end{pmatrix},$$

(2.1)

while the representation for a point $p$ is

$$p^* \equiv \begin{pmatrix} p \\ 1 \end{pmatrix}.$$  (2.2)

Both classes provide a number of methods for applying rotational and affine transforms. Those used for rotations are

```cpp
void transform (R); // this = R * this
void transform (R, v1); // this = R * v1
void inverseTransform (R); // this = inverse(R) * this
void inverseTransform (R, v1); // this = inverse(R) * v1
```

where $R$ is a rotation matrix and $v_1$ is a vector (or a point in the case of `Point3d`).

The methods for applying rigid or affine transforms include:

```cpp
void transform (X); // transforms this by X
void transform (X, v1); // sets this to v1 transformed by X
void inverseTransform (X); // transforms this by the inverse of X
void inverseTransform (X, v1); // sets this to v1 transformed by the inverse of X
```

where $X$ is a rigid or affine transform. As described above, in the case of `Vector3d`, these methods ignore the translational part of the transform and apply only the matrix component ($R$ for a `RigidTransform3d` and $A$ for an `AffineTransform3d`). In particular, that means that for a `RigidTransform3d` given by $X$ and a `Vector3d` given by $v$, the method calls

```cpp
v.transform (X.R)
v.transform (X)
```

produce the same result.

### 2.4 Spatial vectors and inertias

The velocities, forces and inertias associated with 3D coordinate frames and rigid bodies are represented using the 6 DOF spatial quantities described in Sections A.5 and A.6. These are implemented by classes in the package `maspack.spatialmotion`.

Spatial velocities (or twists) are implemented by `Twist`, which exposes its translational and angular velocity components through the publicly accessible fields $v$ and $w$, while spatial forces (or wrenches) are implemented by `Wrench`, which exposes its translational force and moment components through the publicly accessible fields $f$ and $m$.

Both `Twist` and `Wrench` contain methods for algebraic operations such as addition and scaling. They also contain `transform()` methods for applying rotational and rigid transforms. The rotation methods simply transform each component by the supplied rotation matrix. The rigid transform methods, on the other hand, assume that the supplied argument represents a transform between two frames fixed within a rigid body, and transform the twist or wrench accordingly, using either (A.27) or (A.29).

The spatial inertia for a rigid body is implemented by `SpatialInertia`, which contains a number of methods for setting its value given various mass, center of mass, and inertia values, and querying the values of its components. It also contains methods for scaling and adding, transforming between coordinate systems, inversion, and multiplying by spatial vectors.

### 2.5 Meshes

ArtiSynth makes extensive use of 3D meshes, which are defined in `maspack.geometry`. They are used for a variety of purposes, including visualization, collision detection, and computing physical properties (such as inertia or stiffness variation within a finite element model).
A mesh is essentially a collection of vertices (i.e., points) that are topologically connected in some way. All meshes extend the abstract base class `MeshBase`, which supports the vertex definitions, while subclasses provide the topology.

Through `MeshBase`, all meshes provide methods for adding and accessing vertices. Some of these include:

```java
int numVertices(); // return the number of vertices
Vertex3d getVertex (int idx); // return the idx-th vertex
void addVertex (Vertex3d vtx); // add vertex vtx to the mesh
Vertex3d addVertex (Point3d p); // create and return a vertex at position p
void removeVertex (Vertex3d vtx); // remove vertex vtx from the mesh
ArrayList<Vertex3d> getVertices(); // return the list of vertices
```

Vertices are implemented by `Vertex3d`, which defines the position of the vertex (returned by the method `getPosition()`, and also contains support for topological connections. In addition, each vertex maintains an index, obtainable via `getIndex()`, that equals the index of its location within the mesh’s vertex list. This makes it easy to set up parallel array structures for augmenting mesh vertex properties.

Mesh subclasses currently include:

**PolygonalMesh**

Implements a 2D surface mesh containing faces implemented using half-edges.

**PolylineMesh**

Implements a mesh consisting of connected line-segments (polylines).

**PointMesh**

Implements a point cloud with no topological connectivity.

**PolygonalMesh** is used quite extensively and provides a number of methods for implementing faces, including:

```java
int numFaces(); // return the number of faces
Face getFace (int idx); // return the idx-th face
Face addFace (int[] vidxs); // create and add a face using vertex indices
void removeFace (Face f); // remove the face f
ArrayList<Face> getFaces(); // return the list of faces
```

The class `Face` implements a face as a counter-clockwise arrangement of vertices linked together by half-edges (class `HalfEdge`). `Face` also supplies a face’s (outward facing) normal via `getNormal()`.

Some mesh uses within ArtiSynth, such as collision detection, require a *triangular* mesh; i.e., one where all faces have three vertices. The method `isTriangular()` can be used to check for this. Meshes that are not triangular can be made triangular using `triangulate()`.

### 2.5.1 Mesh creation

Meshes are most commonly created using either one of the factory methods supplied by `MeshFactory`, or by reading a definition from a file (Section 2.5.5). However, it is possible to create a mesh by direct construction. For example, the following code fragment creates a simple closed tetrahedral surface:

```java
// a simple four-faced tetrahedral mesh
PolygonalMesh mesh = new PolygonalMesh();
mesh.addVertex (0, 0, 0);
mesh.addVertex (1, 0, 0);
mesh.addVertex (0, 1, 0);
mesh.addVertex (0, 0, 1);
mesh.addFace (new int[] { 0, 2, 1 });
mesh.addFace (new int[] { 0, 3, 2 });
mesh.addFace (new int[] { 0, 1, 3 });
mesh.addFace (new int[] { 1, 2, 3 });
```

Some of the more commonly used factory methods for creating polyhedral meshes include:
MeshFactory.createSphere (radius, nslices, nlevels);
MeshFactory.createIcosahedralSphere (radius, divisions);
MeshFactory.createBox (widthx, widthy, widthz);
MeshFactory.createCylinder (radius, height, nslices);
MeshFactory.createPrism (double[] xycoords, height);
MeshFactory.createTorus (rmajor, rminor, nmajor, nminor);

Each factory method creates a mesh in some standard coordinate frame. After creation, the mesh can be transformed using the transform(X) method, where X is either a rigid transform (RigidTransform3d) or a more general affine transform (AffineTransform3d). For example, to create a rotated box centered on (5, 6, 7), one could do:

```java
// create a box centered at the origin with widths 10, 20, 30:
PolygonalMesh box = MeshFactory.createBox (10, 20, 20);

// move the origin to 5, 6, 7 and rotate using roll-pitch-yaw
// angles 0, 0, 45 degrees:
box.transform(new RigidTransform3d(5, 6, 7, 0, 0, Math.toRadians(45)));
```

One can also scale a mesh using scale(s), where s is a single scale factor, or scale(sx,sy,sz), where sx, sy, and sz are separate scale factors for the x, y and z axes. This provides a useful way to create an ellipsoid:

```java
// start with a unit sphere with 12 slices and 6 levels ...
PolygonalMesh ellipsoid = MeshFactory.createSphere (1.0, 12, 6);

// and then turn it into an ellipsoid by scaling about the axes:
ellipsoid.scale (1.0, 2.0, 3.0);
```

MeshFactory can also be used to create new meshes by performing Boolean operations on existing ones:

```java
MeshFactory.getIntersection (mesh1, mesh2);
MeshFactory.getUnion (mesh1, mesh2);
MeshFactory.getSubtraction (mesh1, mesh2);
```

### 2.5.2 Setting normals, colors, and textures

Meshes provide support for adding normal, color, and texture information, with the exact interpretation of these quantities depending upon the particular mesh subclass. Most commonly this information is used simply for rendering, but in some cases normal information might also be used for physical simulation.

For polygonal meshes, the normal information described here is used only for smooth shading. When flat shading is requested, normals are determined directly from the faces themselves.

Normal information can be set and queried using the following methods:

```java
setNormals {
    List<Vector3d> nrmls, int[] indices); // set all normals and indices
}
ArrayList<Vector3d> getNormals (); // get all normals
int[] getNormalIndices (); // get all normal indices
int numNormals(); // return the number of normals
Vector3d getNormal (int idx); // get the normal at index idx
setNormal (int idx, Vector3d nrml); // set the normal at index idx
clearNormals (); // clear all normals and indices
```

The method setNormals() takes two arguments: a set of normal vectors (nrmls), along with a set of index values (indices) that map these normals onto the vertices of each of the mesh’s geometric features. Often, there will be one unique normal per vertex, in which case nrmls will have a size equal to the number of vertices, but this is not always the case, as described below. Features for the different mesh subclasses are: faces for PolygonalMesh, polylines for
PolylineMesh, and vertices for PointMesh. If indices is specified as null, then normals is assumed to have a size equal to the number of vertices, and an appropriate index set is created automatically using `createVertexIndices()` (described below). Otherwise, indices should have a size equal to the number of features times the number of vertices per feature. For example, consider a PolygonalMesh consisting of two triangles formed from vertex indices (0, 1, 2) and (2, 1, 3), respectively. If normals are specified and there is one unique normal per vertex, then the normal indices are likely to be

\[ 0 \ 1 \ 2 \ 2 \ 1 \ 3 \]

As mentioned above, sometimes there may be more than one normal per vertex. This happens in cases when the same vertex uses different normals for different faces. In such situations, the size of the `nrmls` argument will exceed the number of vertices.

The method `setNormals()` makes internal copies of the specified normal and index information, and this information can be later read back using `getNormals()` and `getNormalIndices()`. The number of normals can be queried using `numNormals()`, and individual normals can be queried or set using `getNormal(idx)` and `setNormal(idx, nrml)`. All normals and indices can be explicitly cleared using `clearNormals()`.

Color and texture information can be set using analogous methods. For colors, we have

```java
setColors (  
    List<float> colors, int[] indices); // set all colors and indices

ArrayList<float> getColors(); // get all colors  
int[] getColorsIndices(); // get all color indices  
int numColors(); // return the number of colors  
float[] getcolor (int idx); // get the color at index idx

setColor (int idx, float[] color); // set the color at index idx  
setColor (int idx, Color color); // set the color at index idx  
setColor (int idx, float r, float g, float b, float a); // set the color at index idx  
clearColors(); // clear all colors and indices
```

When specified as `float[]`, colors are given as RGB or RGBA values, in the range \([0, 1]\), with array lengths of 3 and 4, respectively. The colors returned by `getColors()` are always RGBA values.

With colors, there may often be fewer colors than the number of vertices. For instance, we may have only two colors, indexed by 0 and 1, and want to use these to alternately color the mesh faces. Using the two-triangle example above, the color indices might then look like this:

\[ 0 \ 0 \ 0 \ 1 \ 1 \ 1 \]

Finally, for texture coordinates, we have

```java
setTextureCoords (  
    List<Vector3d> coords, int[] indices); // set all texture coords and indices

ArrayList<Vector3d> getTextureCoords(); // get all texture coords  
int[] getTextureIndices(); // get all texture indices  
int numTextureCoords(); // return the number of texture coords  
Vector3d getTextureCoords (int idx); // get texture coords at index idx

setTextureCoords (int idx, Vector3d coords); // set texture coords at index idx  
clearTextureCoords(); // clear all texture coords and indices
```

When specifying indices using `setNormals`, `setColors`, or `setTextureCoords`, it is common to use the same index set as that which associates vertices with features. For convenience, this index set can be created automatically using

```java
int[] createVertexIndices();
```

Alternatively, we may sometimes want to create a index set that assigns the same attribute to each feature vertex. If there is one attribute per feature, the resulting index set is called a `feature index set`, and can be created using
int[] createFeatureIndices();

If we have a mesh with three triangles and one color per triangle, the resulting feature index set would be

[ 0 0 0 1 1 1 2 2 2 ]

Note: when a mesh is modified by the addition of new features (such as faces for PolygonalMesh), all normal, color and texture information is cleared by default (with normal information being automatically recomputed on demand if automatic normal creation is enabled; see Section 2.5.3). When a mesh is modified by the removal of features, the index sets for normals, colors and textures are adjusted to account for the removal.

For colors, it is possible to request that a mesh explicitly maintain colors for either its vertices or features (Section 2.5.4). When this is done, colors will persist when vertices or features are added or removed, with default colors being automatically created as necessary.

Once normals, colors, or textures have been set, one may want to know which of these attributes are associated with the vertices of a specific feature. To know this, it is necessary to find that feature’s offset into the attribute’s index set. This offset information can be found using the array returned by

int[] getFeatureIndexOffsets()

For example, the three normals associated with a triangle at index ti can be obtained using

int[] indexOffs = mesh.getFeatureIndexOffsets();
ArrayList<Vector3d> nrmls = mesh.getNormals();
// get the three normals associated with the triangle at index ti:
Vector3d n0 = nrmls.get (indexOffs[ti]);
Vector3d n1 = nrmls.get (indexOffs[ti]+1);
Vector3d n2 = nrmls.get (indexOffs[ti]+2);

Alternatively, one may use the convenience methods

Vector3d getFeatureNormal (int idx, int k);
float[] getFeatureColor (int idx, int k);
Vector3d getFeatureTextureCoords (int idx, int k);

which return the attribute values for the k-th vertex of the feature indexed by idx.

In general, the various get methods return references to internal storage information and so should not be modified. However, specific values within the lists returned by getNormals(), getColors(), or getTextureCoords() may be modified by the application. This may be necessary when attribute information changes as the simulation proceeds. Alternatively, one may use methods such as setNormal(idx,nrml) setColor(idx,color), or setTextureCoords(idx,coords).

Also, in some situations, particularly with colors and textures, it may be desirable to not have color or texture information defined for certain features. In such cases, the corresponding index information can be specified as -1, and the getNormal(), getColor() and getTexture() methods will return null for the features in question.

2.5.3 Automatic creation of normals and hard edges

For some mesh subclasses, if normals are not explicitly set, they are computed automatically whenever getNormals() or getNormalIndices() is called. Whether or not this is true for a particular mesh can be queried by the method

boolean hasAutoNormalCreation();

Setting normals explicitly, using a call to setNormals(nrmls,indices), will overwrite any existing normal information, automatically computed or otherwise. The method

boolean hasExplicitNormals();
will return true if normals have been explicitly set, and false if they have been automatically computed or if there is currently no normal information. To explicitly remove normals from a mesh which has automatic normal generation, one may call `setNormals()` with the `nrmls` argument set to null.

More detailed control over how normals are automatically created may be available for specific mesh subclasses. For example, `PolygonalMesh` allows normals to be created with multiple normals per vertex, for vertices that are associated with either open or hard edges. This ability can be controlled using the methods

```java
boolean getMultipleAutoNormals();
setMultipleAutoNormals (boolean enable);
```

Having multiple normals means that even with smooth shading, open or hard edges will still appear sharp. To make an edge hard within a `PolygonalMesh`, one may use the methods

```java
boolean setHardEdge (Vertex3d v0, Vertex3d v1);
boolean setHardEdge (int vidx0, int vidx1);
boolean hasHardEdge (Vertex3d v0, Vertex3d v1);
boolean hasHardEdge (int vidx0, int vidx1);
int numHardEdges();
int clearHardEdges();
```

which control the hardness of edges between individual vertices, specified either directly or using their indices.

### 2.5.4 Vertex and feature coloring

The method `setColors()` makes it possible to assign any desired coloring scheme to a mesh. However, it does require that the user explicitly reset the color information whenever new features are added.

For convenience, an application can also request that a mesh explicitly maintain colors for either its vertices or features. These colors will then be maintained when vertices or features are added or removed, with default colors being automatically created as necessary.

Vertex-based coloring can be requested with the method

```java
setVertexColoringEnabled();
```

This will create a separate (default) color for each of the mesh’s vertices, and set the color indices to be equal to the vertex indices, which is equivalent to the call

```java
setColors (colors, createVertexIndices());
```

where `colors` contains a default color for each vertex. However, once vertex coloring is enabled, the color and index sets will be updated whenever vertices or features are added or removed. Meanwhile, applications can query or set the colors for any vertex using `getColor(idx)`, or any of the various `setColor` methods. Whether or not vertex coloring is enabled can be queried using

```java
getVertexColoringEnabled();
```

Once vertex coloring is established, the application will typically want to set the colors for all vertices, perhaps using a code fragment like this:

```java
mesh.setVertexColoringEnabled();
for (int i=0; i<mesh.numVertices(); i++) {
    ... compute color for the vertex ...
    mesh.setColor (i, color);
}
```

Similarly, feature-based coloring can be requested using the method

```java
setFeatureColoringEnabled();
```

This will create a separate (default) color for each of the mesh’s features (faces for `PolygonalMesh`, polylines for `PolylineMesh`, etc.), and set the color indices to equal the feature index set, which is equivalent to the call
setColors (colors, createFeatureIndices());

where colors contains a default color for each feature. Applications can query or set the colors for any vertex using getColor(idx), or any of the various setColor methods. Whether or not feature coloring is enabled can be queried using getFeatureColoringEnabled();

2.5.5 Reading and writing mesh files

PolygonalMesh, PolylineMesh, and PointMesh all provide constructors that allow them to be created from a definition file, with the file format being inferred from the file name suffix:

<table>
<thead>
<tr>
<th>Suffix</th>
<th>Format</th>
<th>PolygonalMesh</th>
<th>PolylineMesh</th>
<th>PointMesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>.obj</td>
<td>Alias Wavefront</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>.ply</td>
<td>Polygon file format</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>.stl</td>
<td>STereoLithography</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.gts</td>
<td>GNU triangulated surface</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.off</td>
<td>Object file format</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.vtp</td>
<td>VTK ascii format</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.vtp</td>
<td>VTK XML format</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.1: Mesh file formats which are supported for different mesh types

The currently supported file formats, and their applicability to the different mesh types, are given in Table 2.1. For example, a PolygonalMesh can be read from either an Alias Wavefront .obj file or an .stl file, as show in the following example:

```java
 PolygonalMesh mesh0 = null;
 PolygonalMesh mesh1 = null;
 try {
     mesh0 = new PolygonalMesh("meshes/torus.obj");
 } catch (IOException e) {
     System.err.println("Can’t read mesh:");
     e.printStackTrace();
 }
 try {
     mesh1 = new PolygonalMesh("meshes/cylinder.stl");
 } catch (IOException e) {
     System.err.println("Can’t read mesh:");
     e.printStackTrace();
 }
```

The file-based mesh constructors may throw an I/O exception if an I/O error occurs or if the indicated format does not support the mesh type. This exception must either be caught, as in the example above, or thrown out of the calling routine.

In addition to file-based constructors, all mesh types implement read and write methods that allow a mesh to be read from or written to a file, with the file format again inferred from the file name suffix:
For the latter methods, the argument zeroIndexed specifies zero-based vertex indexing in the case of Alias Wavefront .obj files, while fmtStr is a C-style format string specifying the precision and style with which the vertex coordinates should be written. (In the former methods, zero-based indexing is false and vertices are written using full precision.)

As an example, the following code fragment writes a mesh as an .stl file:

```java
PolygonalMesh mesh;
...
initialize ...
try {
    mesh.write (new File("data/mymesh.obj"));
} catch (IOException e) {
    System.err.println("Can’t write mesh:");
    e.printStackTrace();
}
```

Sometimes, more explicit control is needed when reading or writing a mesh from/to a given file format. The constructors and read/write methods described above make use of a specific set of reader and writer classes located in the package maspack.geometry.io. These can be used directly to provide more explicit read/write control. The readers and writers (if implemented) associated with the different formats are given in Table 2.2.

<table>
<thead>
<tr>
<th>Suffix</th>
<th>Format</th>
<th>Reader class</th>
<th>Writer class</th>
</tr>
</thead>
<tbody>
<tr>
<td>.obj</td>
<td>Alias Wavefront</td>
<td>WavefrontReader</td>
<td>WavefrontWriter</td>
</tr>
<tr>
<td>.ply</td>
<td>Polygon file format</td>
<td>PlyReader</td>
<td>PlyWriter</td>
</tr>
<tr>
<td>.stl</td>
<td>Stereolithography</td>
<td>StlReader</td>
<td>StlWriter</td>
</tr>
<tr>
<td>.gts</td>
<td>GNU triangulated surface</td>
<td>GtsReader</td>
<td>GtsWriter</td>
</tr>
<tr>
<td>.off</td>
<td>Object file format</td>
<td>OffReader</td>
<td>OffWriter</td>
</tr>
<tr>
<td>.vtk</td>
<td>VTK ascii format</td>
<td>VtkAsciiReader</td>
<td></td>
</tr>
<tr>
<td>.vtp</td>
<td>VTK XML format</td>
<td>VtkXmlReader</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.2: Reader and writer classes associated with the different mesh file formats

The general usage pattern for these classes is to construct the desired reader or writer with a path to the desired file, and then call readMesh() or writeMesh() as appropriate:

```java
// read a mesh from a .obj file:
WavefrontReader reader = new WavefrontReader("meshes/torus.obj");
PolygonalMesh mesh = null;
try {
    mesh = reader.readMesh();
} catch (IOException e) {
    System.err.println("Can’t read mesh:");
    e.printStackTrace();
}
```

Both readMesh() and writeMesh() may throw I/O exceptions, which must be either caught, as in the example above, or thrown out of the calling routine.

For convenience, one can also use the classes GenericMeshReader or GenericMeshWriter, which internally create an appropriate reader or writer based on the file extension. This enables the writing of code that does not depend on the file format:

```java
String fileName;
...
PolygonalMesh mesh = null;
```
try {
    mesh = (PolygonalMesh) GenericMeshReader.readMesh(fileName);
} catch (IOException e) {
    System.err.println("Can't read mesh:");
    e.printStackTrace();
}

Here, fileName can refer to a mesh of any format supported by GenericMeshReader. Note that the mesh returned by readMesh() is explicitly cast to PolygonalMesh. This is because readMesh() returns the superclass MeshBase, since the default mesh created for some file formats may be different from PolygonalMesh.

2.5.6 Reading and writing normal and texture information

When writing a mesh out to a file, normal and texture information are also written if they have been explicitly set and the file format supports it. In addition, by default, automatically generated normal information will also be written if it relies on information (such as hard edges) that can’t be reconstructed from the stored file information.

Whether or not normal information will be written is returned by the method

    boolean getWriteNormals();

This will always return true if any of the conditions described above have been met. So for example, if a PolygonalMesh contains hard edges, and multiple automatic normals are enabled (i.e., getMultipleAutoNormals() returns true), then getWriteNormals() will return true.

Default normal writing behavior can be overridden within the MeshWriter classes using the following methods:

    int getWriteNormals()  
    setWriteNormals (enable)

where enable should be one of the following values:

0 normals will never be written;  
1 normals will always be written;  
-1 normals will written according to the default behavior described above.

When reading a PolygonalMesh from a file, if the file contains normal information with multiple normals per vertex that suggests the existence of hard edges, then the corresponding edges are set to be hard within the mesh.

2.5.7 Constructive solid geometry

ArtiSynth contains primitives for performing constructive solid geometry (CSG) operations on volumes bounded by triangular meshes. The class that performs these operations is maspack.collision.SurfaceMeshIntersector, and it works by robustly determining the intersection contour(s) between a pair of meshes, and then using these to compute the triangles that need to be added or removed to produce the necessary CSG surface.

The CSG operations include union, intersection, and difference, and are implemented by the following methods of SurfaceMeshIntersector:

    findUnion (mesh0, mesh1);  // volume0 U volume1  
    findIntersection (mesh0, mesh1);  // volume0 ^ volume1  
    findDifference01 (mesh0, mesh1);  // volume0 - volume1  
    findDifference10 (mesh0, mesh1);  // volume1 - volume0

Each takes two PolyhedralMesh objects, mesh0 and mesh1, and creates and returns another PolyhedralMesh which represents the boundary surface of the requested operation. If the result of the operation is null, the returned mesh will be empty.

The example below uses findUnion to create a dumbbell shaped mesh from two balls and a cylinder:
// first create two ball meshes and a bar mesh
double radius = 1.0;
int division = 1; // number of divisions for icosahedral sphere
PolygonalMesh ball0 = MeshFactory.createIcosahedralSphere (radius, division);
ball0.transform (new RigidTransform3d (0, -2*radius, 0));
PolygonalMesh ball1 = MeshFactory.createIcosahedralSphere (radius, division);
ball1.transform (new RigidTransform3d (0, 2*radius, 0));
PolygonalMesh bar = MeshFactory.createCylinder (radius/2, radius*4, /* ns= */32, /* nr= */1, /* nh */10);
bar.transform (new RigidTransform3d (0, 0, 0, 0, Math.PI/2));

// use a SurfaceMeshIntersector to create a CSG union of these meshes
SurfaceMeshIntersector smi = new SurfaceMeshIntersector();
PolygonalMesh balls = smi.findUnion (ball0, ball1);
PolygonalMesh mesh = smi.findUnion (balls, bar);

The balls and cylinder are created using the MeshFactory methods createIcosahedralSphere() and createCylinder(), where the latter takes arguments ns, nr, and nh giving the number of slices along the circumference, end-cap radius, and length. The final resulting mesh is shown in Figure 2.1.

### 2.6 Reading source relative files

ArtiSynth applications frequently need to read in various kinds of data files, including mesh files (as discussed in Section 2.5.5), FEM mesh geometry (Section 6.2.2), probe data (Section 5.4.4), and custom application data.

Often these data files do not reside in an absolute location but instead in a location relative to the application’s class or source file. For example, it is common for applications to store geometric data in a subdirectory "geometry" located beneath the source directory. In order to access such files in a robust way, and ensure that the code does not break when the source tree is moved, it is useful to determine the application’s source (or class) directory at run time. ArtiSynth supplies several ways to conveniently handle this situation. First, the RootModel itself supplies the following methods:

```java
// find path to the root model's source directory
String findSourceDir ();

// get path to a file specified relative to the root model's source directory
String getSourceRelativePath (String relpath);
```

The first method returns the path to the source directory of the root model, while the second returns the path to a file specified relative to the root model source directory. If the root model source directory cannot be found (see discussion at the end of this section) both methods return null. As a specific usage example, assume that we have an application model whose build() method needs to load in a mesh torus.obj from a subdirectory meshes located beneath the source directory. This could be done as follows:
String pathToMesh = getSourceRelativePath("meshes/torus.obj");

// read the mesh from a .obj file :
WavefrontReader reader = new WavefrontReader(pathToMesh);
PolygonalMesh mesh = null;
try {
    mesh = reader.readMesh();
} catch (IOException e) {
    System.err.println("Can’t read mesh:");
    e.printStackTrace();
}

A more general path finding utility is provided by maspack.util.PathFinder, which provides several static methods for locating source and class directories:

// find path to the source directory associated with classObj
String findSourceDir(Object classObj);

// get path to a file specified relative to classObj source directory
String getSourceRelativePath(Object classObj, String relpath);

// find path to the class directory associated with classObj
String findClassDir(Object classObj);

// get path to a file specified relative to classObj class directory
String getClassRelativePath(Object classObj, String relpath);

The “find” methods return a string path to the indicated class or source directory, while the “relative path” methods locate the class or source directory and append the additional path relpath. For all of these, the class is determined from classObj, either directly (if it is an instance of Class), by name (if it is a String), or otherwise by calling classObj.getClass(). When identifying a package by name, the name should be either a fully qualified class name, or a simple name that can be located with respect to the packages obtained via Package.getPackages(). For example, if we have a class whose fully qualified name is artisynth.models.test.Foo, then the following calls should all return the same result:

Foo foo = new Foo();
PathFinder.findSourceDir(foo);
PathFinder.findSourceDir(Foo.class);
PathFinder.findSourceDir("artisynth.models.test Foo");
PathFinder.findSourceDir("Foo");

If the source directory for Foo happens to be /home/projects/src/artisynth/models/test, then

PathFinder.getSourceRelativePath (foo, "geometry/mesh.obj");

will return /home/projects/src/artisynth/models/test/geometry/mesh.obj.

When calling PathFinder methods from within the relevant class, one can specify this as the classObj argument.

With respect to the above example locating the file "meshes/torus.obj", the call to the root model method
getSourceRelativePath() could be replaced with

String pathToMesh = PathFinder.getSourceRelativePath {
    this, "meshes/torus.obj");

Since this is assumed to be called from the root model’s build method, the “class” can be indicated by simply passing this to getSourceRelativePath().
As an alternative to placing data files in the source directory, one could place them in the class directory, and then use `findClassDir()` and `getClassRelativePath()`. If the data files were originally defined in the source directory, it will be necessary to copy them to the class directory. Some Java IDEs will perform this automatically.

The `PathFinder` methods work by climbing the class’s resource hierarchy. Source directories are assumed to be located relative to the parent of the root class directory, via one of the paths specified by `getSourceRootPaths()`. By default, this list includes "src", "source", and "bin". Additional paths can be added using `addSourceRootPath(path)`, or the entire list can be set using `setSourceRootPaths(paths)`.

At present, source directories will not be found if the reference class is contained in a jar file.

### 2.7 Reading and caching remote files

ArtiSynth applications often require the use of large data files to specify items such as FEM mesh geometry, surface mesh geometry, or medical imaging data. The size of these files may make it inconvenient to store them in any version control system that is used to store the application source code. As an alternative, ArtiSynth provides a file manager utility that allows such files to be stored on a separate server, and then downloaded on-demand and cached locally. To use this, one starts by creating an instance of a `FileManager`, using the constructor

```java
FileManager (String downloadPath, String remoteSourceName)
```

where `downloadPath` is a path to the local directory where the downloaded file should be placed, and `remoteSourceName` is a URI indicating the remote server location of the files. After the file manager has been created, it can be used to fetch remote files and cache them locally, using various `get` methods:

```java
File get (String destName);
File get (String destName, String sourceName);
```

Both of these look for the file `destName` specified relative to the local directory, and return a `File` handle for it if it is present. Otherwise, they attempt to download the file from the remote source location, place it in the local directory, and return a `File` handle for it. The location of the remote file is given relative to the remote source URI by `destName` for the first method and `sourceName` for the second.

A simple example of using a file manager within a `RootModel build()` method is given by the following fragment:

```java
// create the file manager ...
FileManager fm = new FileNameManager {
   getSourceRelativePath("geometry"),
   "http://myserver.org/artisynth/data/geometry";
// ... and use it to get a bone mesh file
File meshFile = fm.get("tibia.obj");
```

Here, a file manager is created that uses a local directory "geometry", located relative to the `RootModel source` directory (see Section 2.6), and looks for missing files relative to the URI

```text
http://myserver.org/artisynth/data/geometry
```

The `get()` method is then used to obtain the file "tibia.obj" from the local directory. If it is not already present, it is downloaded from the remote location.

The `FileManager` contains other features and functionality, and one should consult its API documentation for more information.
Chapter 3
Mechanical Models I

This section details how to build basic multibody-type mechanical models consisting of particles, springs, rigid bodies, joints, and other constraints.

3.1 Springs and particles

The most basic type of mechanical model consists simply of particles connected together by axial springs. Particles are implemented by the class `Particle`, which is a dynamic component containing a three-dimensional position state, a corresponding velocity state, and a mass. It is an instance of the more general base class `Point`, which is used to also implement spatial points such as markers which do not have a mass.

3.1.1 Axial springs and materials

An axial spring is a simple spring that connects two points and is implemented by the class `AxialSpring`. This is a force effector component that exerts equal and opposite forces on the two points, along the line separating them, with a magnitude $f$ that is a function $f(l, \dot{l})$ of the distance $l$ between the points, and the distance derivative $\dot{l}$.

Each axial spring is associated with an axial material, implemented by a subclass of `AxialMaterial`, that specifies the function $f(l, \dot{l})$. The most basic type of axial material is a `LinearAxialMaterial`, which determines $f$ according to the linear relationship

$$f(l, \dot{l}) = k(l - l_0) + d\dot{l}$$

where $l_0$ is the rest length and $k$ and $d$ are the stiffness and damping terms. Both $k$ and $d$ are properties of the material, while $l_0$ is a property of the spring.

Axial springs are assigned a linear axial material by default. More complex, nonlinear axial materials may be defined in the package `artisynth.core.materials`. Setting or querying a spring’s material may be done with the methods `setMaterial()` and `getMaterial()`.

3.1.2 Example: a simple particle-spring model

An complete application model that implements a simple particle-spring model is given below.

```java
package artisynth.demos.tutorial;

import java.awt.Color;
import maspack.matrix.*;
import maspack.render.*;
import artisynth.core.mechmodels.*;
import artisynth.core.materials.*;
import artisynth.core.workspace.RootModel;

/**

 */
```
Figure 3.1: ParticleSpring model loaded into ArtiSynth.

* Demo of two particles connected by a spring

```java
public class ParticleSpring extends RootModel {
    public void build(String[] args) {
        // create MechModel and add to RootModel
        MechModel mech = new MechModel("mech");
        addModel(mech);

        // create the components
        Particle p1 = new Particle("p1", /* mass=*/2, /*x,y,z=*/0, 0, 0);
        Particle p2 = new Particle("p2", /* mass=*/2, /*x,y,z=*/1, 0, 0);
        AxialSpring spring = new AxialSpring("spr", /* restLength=*/0);
        spring.setPoints(p1, p2);
        spring.setMaterial("stiffness=*/20, /* damping=*/10);

        // add components to the mech model
        mech.addParticle(p1);
        mech.addParticle(p2);
        mech.addAxialSpring(spring);

        p1.setDynamic(false); // first particle set to be fixed

        // increase model bounding box for the viewer
        mech.setBounds("min=*/-1, 0, -1, /*max=*/1, 0, 0);
        // set render properties for the components
        RenderProps.setSphericalPoints(p1, 0.06, Color.RED);
        RenderProps.setSphericalPoints(p2, 0.06, Color.RED);
        RenderProps.setCylindricalLines(spring, 0.02, Color.BLUE);
    }
}
```

Line 1 of the source defines the package in which the model class will reside, in this case artisynth.demos.tutorial. Lines 3-8 import definitions for other classes that will be used.

The model application class is named ParticleSpring and declared to extend RootModel (line 13), and the build() method definition begins at line 15. (A no-args constructor is also needed, but because no other constructors are defined, the compiler creates one automatically.)

To begin, the build() method creates a MechModel named "mech", and then adds it to the models list of the root model.
using the `addModel()` method (lines 18–19). Next, two particles, `p1` and `p2`, are created, with masses equal to 2 and initial positions at 0, 0, 0, and 1, 0, 0, respectively (lines 22–23). Then an axial spring is created, with end points set to `p1` and `p2`, and assigned a linear material with a stiffness and damping of 20 and 10 (lines 24–27). Finally, after the particles and the spring are created, they are added to the `particles` and `axialSprings` lists of the `MechModel` using the methods `addParticle()` and `addAxialSpring()` (lines 30–32).

At this point in the code, both particles are defined to be dynamically controlled, so that running the simulation would cause both to fall under the `MechModel`’s default gravity acceleration of (0, 0, −9.8). However, for this example, we want the first particle to remain fixed in place, so we set it to be non-dynamic (line 34), meaning that the physical simulation will not update its position in response to forces (Section 3.1.3).

The remaining calls control aspects of how the model is graphically rendered. `setBounds()` (line 37) increases the model’s “bounding box” so that by default it will occupy a larger part of the viewer frustum. The convenience method `RenderProps.setSphericalPoints()` is used to set points `p1` and `p2` to render as solid red spheres with a radius of 0.06, while `RenderProps.setCylindricalLines()` is used to set spring to render as a solid blue cylinder with a radius of 0.02. More details about setting render properties are given in Section 4.3.

To run this example in ArtiSynth, select All demos > tutorial > ParticleSpring from the Models menu. The model should load and initially appear as in Figure 3.1. Running the model (Section 1.5.3) will cause the second particle to fall and swing about under gravity.

### 3.1.3 Dynamic, parametric, and attached components

By default, a dynamic component is advanced through time in response to the forces applied to it. However, it is also possible to set a dynamic component’s dynamic property to false, so that it does not respond to force inputs. As shown in the example above, this can be done using the method `setDynamic()`:

```java
    comp.setDynamic (false);
```

The method `isDynamic()` can be used to query the dynamic property.

Dynamic components can also be attached to other dynamic components (as mentioned in Section 1.2) so that their positions and velocities are controlled by the master components that they are attached to. To attach a dynamic component, one creates an `AttachmentComponent` specifying the attachment connection and adds it to the ` MechModel`, as described in Section 3.6. The method `isAttached()` can be used to determine if a component is attached, and if it is, `getAttachment()` can be used to find the corresponding `AttachmentComponent`.

Overall, a dynamic component can be in one of three states:

- **active**: Component is dynamic and unattached. The method `isActive()` returns `true`. The component will move in response to forces.

- **parametric**: Component is not dynamic, and is unattached. The method `isParametric()` returns `true`. The component will either remain fixed, or will move around in response to external inputs specifying the component’s position and/or velocity. One way to supply such inputs is to use controllers or input probes, as described in Section 5.

- **attached**: Component is attached. The method `isAttached()` returns `true`. The component will move so as to follow the other master component(s) to which it is attached.

### 3.1.4 Custom axial materials

Application authors may create their own axial materials by subclassing `AxialMaterial` and overriding the functions:

```java
    double computeF (l, ldot, l0, excitation);
    double computeDFdl (l, ldot, l0, excitation);
    double computeDFdldot (l, ldot, l0, excitation);
    boolean isDFdldotZero ();
```
where excitation is an additional excitation signal \( a \), which is used to implement active springs and which in particular is used to implement axial muscles (Section 4.4), for which \( a \) is usually in the range \([0, 1]\).

The first three methods should return the values of

\[
\begin{align*}
    f(l, l', a), \\ 
    \frac{\partial f(l, l', a)}{\partial l'}, \\ 
    \frac{\partial f(l, l', a)}{\partial l},
\end{align*}
\]

respectively, while the last method should return \( \text{true} \) if \( \frac{\partial f(l, l', a)}{\partial l'} \equiv 0 \); i.e., if it is always equals to 0.

### 3.1.5 Damping parameters

Mechanical models usually contain damping forces in addition to spring-type restorative forces. Damping generates forces that reduce dynamic component velocities, and is usually the major source of energy dissipation in the model. Damping forces can be generated by the spring components themselves, as described above.

A general damping can be set for all particles by setting the MechModel’s `pointDamping` property. This causes a force

\[
f_i = -d_p v_i
\]

(3.3)
to be applied to all particles, where \( d_p \) is the value of the `pointDamping` and \( v_i \) is the particle’s velocity.

`pointDamping` can be set and queried using the MechModel methods

```java
setPointDamping(double d);
double getPointDamping();
```

In general, whenever a component has a property `propX`, that property can be set and queried in code using methods of the form

```java
setPropX(T d);
T getPropX();
```

where \( T \) is the type associated with the property.

`pointDamping` can also be set for particles individually. This property is inherited (Section 1.4.3), so that if not set explicitly, it inherits the nearest explicitly set value in an ancestor component.

### 3.2 Rigid bodies

Rigid bodies are implemented in ArtiSynth by the class `RigidBody`, which is a dynamic component containing a six-dimensional position and orientation state, a corresponding velocity state, an inertia, and an optional surface mesh.

A rigid body is associated with its own 3D spatial coordinate frame, and is a subclass of the more general `Frame` component. The combined position and orientation of this frame with respect to world coordinates defines the body’s pose, and the associated 6 degrees of freedom describe its “position” state.

#### 3.2.1 Frame markers

ArtiSynth makes extensive use of markers, which are (massless) points attached to dynamic components in the model. Markers are used for graphical display, implementing attachments, and transmitting forces back onto the underlying dynamic components.

A frame marker is a marker that can be attached to a `Frame`, and most commonly to a `RigidBody` (Figure 3.2). They are frequently used to provide the anchor points for attaching springs and, more generally, applying forces to the body.

Frame markers are implemented by the class `FrameMarker`, which is a subclass of `Point`. The methods

```java
Point3d getLocation();
void setLocation(Point3d r);
```
get and set the marker’s location \( \mathbf{r} \) with respect to the frame’s coordinate system. When a 3D force \( \mathbf{f} \) is applied to the marker, it generates a spatial force \( \hat{\mathbf{f}} \) (Section A.5) on the frame given by

\[
\hat{\mathbf{f}} = \left( \mathbf{f} \times \mathbf{r} \right).
\]

(3.4)

Frame markers can be created using a variety of constructors, including

```java
FrameMarker();
FrameMarker(String name);
FrameMarker(Frame frame, Point3d loc);
```

where `FrameMarker()` creates an empty marker, `FrameMarker(name)` creates an empty marker with a name, and `FrameMarker(frame, loc)` creates an unnamed marker attached to `frame` at the location `loc` with respect to the frame’s coordinates. Once created, a marker’s frame can be set and queried with

```java
void setFrame (Frame frame);
Frame getFrame () ;
```

A frame marker can be added to a `MechModel` with the `MechModel` methods

```java
void addFrameMarker (FrameMarker mkr);  
void addFrameMarker (FrameMarker mkr, Frame frame, Point3d loc);  
```

where `addFrameMarker(mkr, frame, loc)` also sets the frame and the marker’s location with respect to it.

`MechModel` also supplies convenience methods to create a marker, attach it to a frame, and add it to the model:

```java
FrameMarker addFrameMarker (Frame frame, Point3d loc);  
FrameMarker addFrameMarkerWorld (Frame frame, Point3d locw);  
```

Both methods return the created marker. The first, `addFrameMarker(frame, loc)`, places it at the location `loc` with respect to the frame, while `addFrameMarkerWorld(frame, pos)` places it at `pos` with respect to `world` coordinates.

### 3.2.2 Example: a simple rigid body-spring model

A simple rigid body-spring model is defined in

```
artisynth.demos.tutorial.RigidBodySpring
```

This differs from `ParticleSpring` only in the `build()` method, which is listed below:
```java
public void build (String[] args) {

    // create MechModel and add to RootModel
    MechModel mech = new MechModel("mech");
    addModel (mech);

    // create the components
    Particle p1 = new Particle("p1", /* mass=*/2, /* x, y, z *//0, 0, 0);
    // create box and set its pose (position/orientation):
    RigidBody box =
        RigidBody.createBox("box", /* wx, wy, wz *//0.5, 0.3, 0.3, /* density=*//20);
    box.setPose (new RigidTransform3d(/* x, y, z *//0.75, 0, 0));
    // create marker point and connect it to the box:
    FrameMarker mkr = new FrameMarker(/* x, y, z *//-0.25, 0, 0);
    mkr setFrame (box);

    AxialSpring spring = new AxialSpring("spr", /* restLength=*//0);
    spring.setPoints (p1, mkr);
    spring.setMaterial (new LinearAxialMaterial (/* stiffness=*//20, /* damping=*//10));

    // add components to the mech model
    mech.addParticle (p1);
    mech.addRigidBody (box);
    mech.addFrameMarker (mkr);
    mech.addAxialSpring (spring);

    p1.setDynamic (false);  // first particle set to be fixed

    // increase model bounding box for the viewer
    mech.setBounds (/* min=*//−1, 0, −1, /* max=*//1, 0, 0);

    // set render properties for the components
    RenderProps.setSphericalPoints (p1, 0.06, Color.RED);
    RenderProps.setSphericalPoints (mkr, 0.06, Color.RED);
    RenderProps.setCylindricalLines (mkr, 0.02, Color.BLUE);
}
```

The differences from ParticleSpring begin at line 9. Instead of creating a second particle, a rigid body is created using the factory method RigidBody.createBox(), which takes x, y, z widths and a (uniform) density and creates a box-shaped rigid body complete with surface mesh and appropriate mass and inertia. As the box is initially centered at the
origin, moving it elsewhere requires setting the body’s pose, which is done using setPose(). The RigidTransform3d passed to setPose() is created using a three-argument constructor that generates a translation-only transform. Next, starting at line 14, a FrameMarker is created for a location \((-0.25, 0, 0)^T\) relative to the rigid body, and attached to the body using its setFrame() method.

The remainder of build() is the same as for ParticleSpring, except that the spring is attached to the frame marker instead of a second particle.

To run this example in ArtiSynth, select All demos > tutorial > RigidBodySpring from the Models menu. The model should load and initially appear as in Figure 3.3. Running the model (Section 1.5.3) will cause the rigid body to fall and swing about under gravity.

### 3.2.3 Creating rigid bodies

As illustrated above, rigid bodies can be created using factory methods supplied by RigidBody. Some of these include:

- `createBox(name, widthX, widthY, widthZ, density);`
- `createCylinder(name, radius, height, density, nsides);`
- `createSphere(name, radius, density, nslices);`
- `createEllipsoid(name, radX, radY, radZ, density, nslices);`

The bodies do not need to be named; if no name is desired, then name and can be specified as `null`.

In addition, there are also factory methods for creating a rigid body directly from a mesh:

- `createFromMesh(name, mesh, density, scale);`
- `createFromMesh(name, meshFileName, density, scale);`

These take either a polygonal mesh (Section 2.5), or a file name from which a mesh is read, and use it as the body’s surface mesh and then compute the mass and inertia properties from the specified (uniform) density.

When a body is created directly from a surface mesh, its center of mass will typically *not* be coincident with the origin of its coordinate frame. Section 3.2.6 discusses the implications of this and how to correct it.

Alternatively, one can create a rigid body directly from a constructor, and then set the mesh and inertia properties explicitly:

```java
PolygonalMesh femurMesh;
SpatialInertia inertia;
...
initialize mesh and inertia appropriately ...

RigidBody body = new RigidBody("femur");
body.setMesh(femurMesh);
body.setInertia(inertia);
```

### 3.2.4 Pose and velocity

A body’s pose can be set and queried using the methods

```java
setPose(RigidTransform3d T); // sets the pose to T
getPose(RigidTransform3d T); // gets the current pose in T
RigidTransform3d getPose(); // returns the current pose (read-only)
```

These use a RigidTransform3d (Section 2.2) to describe the pose. Body poses are described in world coordinates and specify the transform from body to world coordinates. In particular, the pose for a body A specifies the rigid transform \(T_{AW}\).

Rigid bodies also expose the translational and rotational components of their pose via the properties `position` and `orientation`, which can be queried and set independently using the methods
The velocity of a rigid body is described using a **Twist** (Section 2.4), which contains both the translational and rotational velocities. The following methods set and query the spatial velocity as described with respect to world coordinates:

```java
setPosition (Point3d p); // sets the position to p
getPosition (Point3d p); // gets the current position in p
Point3d getPosition(); // returns the current position (read-only)

setOrientation (AxisAngle a); // sets the orientation to a
getOrientation (AxisAngle a); // gets the current orientation in a
AxisAngle getOrientation(); // returns the current orientation (read-only)
```

During simulation, unless a rigid body has been set to be **parametric** (Section 3.1.3), its pose and velocity are updated in response to forces, so setting the pose or velocity generally makes sense only for setting initial conditions. On the other hand, if a rigid body is parametric, then it is possible to control its pose during the simulation, but in that case it is better to set its **target pose** and/or **target velocity**, as described in Section 5.3.1.

### 3.2.5 Inertia and the surface mesh

The “mass” of a rigid body is described by its spatial inertia, which is a $6 \times 6$ matrix relating its spatial velocity to its spatial momentum (Section A.6). Within ArtiSynth, spatial inertia is described by a **SpatialInertia** object, which specifies its mass, center of mass (with respect to body coordinates), and rotational inertia (with respect to the center of mass).

Most rigid bodies are also associated with a polygonal surface mesh, which can be set and queried using the methods

```java
setSurfaceMesh (PolygonalMesh mesh); // set SurfaceMesh
setSurfaceMesh (PolygonalMesh mesh, String meshFileName); // set SurfaceMesh and fileName
PolygonalMesh getSurfaceMesh(); // return SurfaceMesh
```

The second method takes an optional `fileName` argument that can be set to the name of a file from which the mesh was read. Then if the model itself is saved to a file, the model file will specify the mesh using the file name instead of explicit vertex and face information, which can reduce the model file size considerably.

Rigid bodies can also have more than one mesh, as described in Section 3.2.9.

The inertia of a rigid body can be explicitly set using a variety of methods including

```java
setInertia (M) // set using SpatialInertia M
setInertia (mass, Jxx, Jyy, Jzz); // mass and diagonal rotational inertia
setInertia (mass, J); // mass and full rotational inertia
setInertia (mass, J, com); // mass, rotational inertia, center-of-mass
```

and can be queried using

```java
getInertia (M); // get SpatialInertia in M
getInertia (); // return read-only SpatialInertia
```

In practice, it is often more convenient to simply specify a mass or a density, and then use the geometry of the surface mesh (and possibly other meshes, Section 3.2.9) to compute the remaining inertial values. How a rigid body’s inertia is computed is determined by its `inertiaMethod` property, which can be one of:

**EXPLICIT**

Inertia is set explicitly.

**MASS**

Inertia is determined implicitly from the mesh geometry and the body’s mass.
DENSITY

Inertia is determined implicitly from the mesh geometry and the body’s density (which is multiplied by the mesh volume(s) to determine a mass).

When using DENSITY to determine the inertia, it is generally assumed that the contributing meshes are both polygonal and closed. Meshes which are either open or non-polygonal generally do not have a well-defined volume which can be multiplied by the density to determine the mass.

The inertiaMethod property can be set and queried using

```
setInertiaMethod (InertiaMethod method);
InertiaMethod getInertiaMethod ();
```

and its default value is DENSITY. Explicitly setting the inertia using one of setInertia() methods described above will set inertiaMethod to EXPLICIT. The method

```
setInertiaFromDensity (density);
```

will (re)compute the inertia using the mesh geometry and a density value and set inertiaMethod to DENSITY, and the method

```
setInertiaFromMass (mass);
```

will (re)compute the inertia using the mesh geometry and a mass value and set inertiaMethod to MASS.

Finally, the (assumed uniform) density of the body can be queried using

```
getDensity ();
```

There are some subtleties involved in determining the inertia using either the DENSITY or MASS methods when the rigid body contains more than one mesh. Details are given in Section 3.2.9.

3.2.6 Coordinate frames and the center of mass

Figure 3.4: Left: rigid body whose coordinate frame B is not coincident with the center of mass (COM). Right: same body, with its coordinate frame translated to be coincident with the COM.

It is important to note that the origin of a body’s coordinate frame will not necessarily coincide with its center of mass (COM), and in fact the frame origin does not even have to lie inside the body’s surface (Figure 3.4). This typically occurs when a body’s inertia is computed directly from its surface mesh (or meshes), as described in Section 3.2.5.

Having the COM differ from the frame origin may lead to some undesired effects. For instance, since the body’s spatial velocity is defined with respect to the frame origin and not the COM, if the two are not coincident, then a purely angular body velocity will cause the COM to translate. The body’s spatial inertia also becomes more complicated, with non-zero
3 x 3 blocks in the lower left and upper right (Section A.6), which can have a small effect on computational accuracy. Finally, manipulating a body’s pose in the ArtiSynth UI (as described in the section “Model Manipulation” in the ArtiSynth User Interface Guide) can also be more cumbersome if the origin is located far from the COM.

There are several ways to ensure that the COM and frame origin are coincident. The most direct is to call the method centerPoseOnCenterOfMass() after the body has been created:

```java
String meshFilePath = "/#project/geometry/bodyMesh.obj";
double density = 1000;

PolygonalMesh mesh = new PolygonalMesh(meshFilePath); // read in a mesh
RigidBody bodyA = RigidBody.createFromMesh("bodyA", mesh, density, /* scale=*/1); // create body from the mesh
bodyA.centerPoseOnCenterOfMass(); // center body on the COM
```

This will shift the body’s frame to be coincident with the COM, while at the same time translating its mesh vertices in the opposite direction so that its mesh (or meshes) don’t move with respect to world coordinates. The spatial inertia is updated as well.

Alternatively, if the body is being created from a single mesh, one may transform that mesh to be centered on its COM before it is used to define the body. This can be done using the PolygonalMesh method translateToCenterOfVolume(), which centers a mesh’s vertices on its COM (assuming a uniform density):

```java
PolygonalMesh mesh = new PolygonalMesh(meshFilePath); // read in a mesh
mesh.translateToCenterOfVolume(); // center mesh on its COM
RigidBody bodyA = RigidBody.createFromMesh("bodyA", mesh, density, /* scale=*/1); // create body from the mesh
```

### 3.2.7 Damping parameters

As with particles, it is possible to set damping parameters for rigid bodies. This can be specified in two different ways:

1. **Translational/rotational** damping which is proportional to a body’s translational and rotational velocity;
2. **Inertial** damping, which is proportional to a body’s spatial velocity multiplied by its spatial inertia.

Translational/rotational damping is controlled by the MechModel properties frameDamping and rotaryDamping, and generates a spatial force centered on each rigid body’s coordinate frame given by

\[
\hat{f} = -\begin{pmatrix} d_f v \\ d_r \omega \end{pmatrix},
\]

where \(d_f\) and \(d_r\) are the frameDamping and rotaryDamping values, and \(v\) and \(\omega\) are the translational and angular velocity of the body’s coordinate frame. The damping parameters can be set and queried using the MechModel methods:

```java
setFrameDamping (double df)
setRotaryDamping (double dr)
double getFrameDamping ()
double getRotaryDamping ()
```

These damping parameters can also be set for individual bodies using their own (inherited) frameDamping and rotary-Damping properties.

For models involving rigid bodies, it is often necessary to set rotaryDamping to a non-zero value because frameDamping will provide no damping at all when a rigid body is simply rotating about its coordinate frame origin.

Inertial damping is controlled by the MechModel property inertialDamping, and generates a spatial force centered on a rigid body’s coordinate frame given by

\[
\hat{f} = -d_I M \hat{v}, \quad \hat{v} \equiv \begin{pmatrix} v \\ \omega \end{pmatrix},
\]

where \(d_I\) is the inertialDamping, \(M\) is the body’s 6 \times 6 spatial inertia matrix (Section A.6), and \(\hat{v}\) is the body’s spatial velocity. The inertial damping property can be set and queried using the MechModel methods:
setInertialDamping (double di)
double getInertialDamping()

This parameter can also be set for individual bodies using their own (inherited) inertialDamping property.

Inertial damping offers two advantages over translational/rotational damping:

1. It is independent of the location of the body’s coordinate frame with respect to its center of mass;
2. There is no need to adjust two different translational and rotational parameters or to consider their relative sizes, as these considerations are contained within the spatial inertia itself.

3.2.8 Rendering rigid bodies

A rigid body is rendered in ArtiSynth by drawing its mesh (or meshes, Section 3.2.9) and/or coordinate frame.

Meshes are drawn using the face rendering properties described in more detail in Section 4.3. The most commonly used of these are:

- **faceColor**: A value of type `java.awt.Color` giving the color of mesh faces. The default value is `GRAY`.
- **shading**: A value of type `Renderer.Shading` indicating how the mesh should be shaded, with the options being `FLAT`, `SMOOTH`, `METAL`, and `NONE`. The default value is `FLAT`.
- **alpha**: A double value between 0 and 1 indicating transparency, with transparency increasing as value decreases from 1. The default value is 1.
- **faceStyle**: A value of type `Renderer.FaceStyle` indicating which face sides should be drawn, with the options being `FRONT`, `BACK`, `FRONT_AND_BACK`, and `NONE`. The default value is `FRONT`.
- **drawEdges**: A boolean indicating whether the mesh edges should also be drawn, using either the `edgeColor` rendering property, or the `lineColor` property if `edgeColor` is not set. The default value is `false`.

These properties, and others, can be set either interactively in the GUI, or in code. To set the render properties in the GUI, select the rigid body or its mesh component, and then right click the mouse and choose `Edit render props ...`. More details are given in the section “Render properties” in the *ArtiSynth User Interface Guide*.

Figure 3.5: Different rendering settings for a rigid body hip mesh showing the default (left), smooth rendering with a lighter color (center), and wireframe (right).

Properties can also be set in code, usually during the `build()` method. Typically this is done using a static method of the `RenderProps` class that has the form

```java
RenderProps.setXXX (comp, value)
```
where XXX is the property name, comp is the component for which the property should be set, and value is the desired value. Some examples are shown in Figure 3.5 for a rigid body hip representation with a fairly coarse mesh. The left image shows the default rendering, using a gray color and flat shading. The center image shows a lighter color and smooth shading, which could be set by the following code fragment:

```java
import maspack.render.*;
import maspack.render.Renderer.*;
...
RigidBody hipBody;
...
RenderProps.setFaceColor (hipBody, new Color (255, 255, 204));
RenderProps.setShading (hipBody, Shading.SMOOTH);
```

Finally, the right image shows the body rendered as a wire frame, which can be done by setting faceStyle to NONE and drawEdges to true:

```java
RenderProps.setFaceStyle (hip, FaceStyle.NONE);
RenderProps.setDrawEdges (hip, true);
RenderProps.setEdgeWidth (hip, 2);
RenderProps.setEdgeColor (hip, Color.CYAN);
```

Render properties can also be set in higher level model components, from which their values will be inherited by lower level components that have not explicitly set their own values. For example, setting the faceColor render property in the MechModel will automatically set the face color for all subcomponents which have not explicitly set faceColor. More details on render properties are given in Section 4.3.

![Figure 3.6: Rigid body axes rendered with axisDrawStyle set to LINE (left) and ARROW (right).](image)

In addition to mesh rendering, it is often useful to draw a rigid body’s coordinate frame, which can be done using its axisLength and axisDrawStyle properties. Setting axisLength to a positive value will cause the body’s three coordinate axes to be drawn, with the indicated length, with the x, y and z axes colored red, green, and blue, respectively. The axisDrawStyle property controls how the axes are rendered (Figure 3.6). It has the type `Renderer.AxisDrawStyle`, and can be set to the following values:

**OFF**

Axes are not rendered.

**LINE**

Axes are rendered as simple red-green-blue lines, with a width given by the joint’s lineWidth rendering property.

**ARROW**

Axes are rendered as solid red-green-blue arrows.
As with the rendering properties, the `axisLength` and `axisDrawStyle` properties can be managed either interactively in the GUI (by selecting the body, right clicking and choosing Edit properties ...), or in code, using the following methods:

```java
double getAxisLength()
void setAxisLength(double len)
AxisDrawStyle getAxisDrawStyle()
void setAxisDrawStyle(AxisDrawStyle style)
```

### 3.2.9 Multiple meshes

A `RigidBody` may contain multiple meshes, which can be useful for various reasons:

- It may be desirable to use different meshes for collision detection, inertia computation, and visual presentation;
- Different render properties can be set for different mesh components, allowing the body to be rendered in a more versatile way;
- Different mesh components can be selected individually.

Each rigid body mesh is encapsulated inside a `RigidMeshComp` component, which is in turn stored in a subcomponent list called `meshes`. Meshes do not need to be instances of `PolygonalMesh`; instead, they can be any instance of `MeshBase`, including `PointMesh` and `PolylineMesh`.

The default surface mesh, returned by `getSurfaceMesh()`, is also stored inside a `RigidMeshComp` in the `meshes` list. By default, the surface mesh is the first mesh in the list, but is otherwise defined to be the first mesh in `meshes` which is also an instance of `PolygonalMesh`. The `RigidMeshComp` containing the surface mesh can be obtained using the method `getSurfaceMeshComp()`.

A `RigidMeshComp` contains a number of properties that control how the mesh is displayed and interacts with its rigid body:

- **renderProps**
  Render properties controlling how the mesh is rendered (see Section 4.3).

- **hasMass**
  A boolean, which if `true` means that the mesh will contribute to the body’s inertia when the `inertiaMethod` is either `MASS` or `DENSITY`. The default value is `true`.

- **massDistribution**
  An enumerated type defined by `MassDistribution` which specifies how the mesh’s inertia contribution is determined for a given mass. `VOLUME`, `AREA`, `LENGTH`, and `POINT` indicate, respectively, that the mass is distributed evenly over the mesh’s volume, area (faces), length (edges), or points. The default value is determined by the mesh type: `VOLUME` for a closed `PolygonalMesh`, `AREA` for an open `PolygonalMesh`, `LENGTH` for a `PolylineMesh`, and `POINT` for a `PointMesh`. Applications can specify an alternate value providing the mesh has the features to support it. Specifying `DEFAULT` will restore the default value.

- **isCollidable**
  A boolean, which if `true`, and if the mesh is a `PolygonalMesh`, means that the mesh will take part in collision and wrapping interactions (Chapter 7 and Section 8.3). The default value is `true`, and the get/set accessors have the names `isCollidable()` and `setIsCollidable()`.

- **volume**
  A double whose value is the volume of the mesh. If the mesh is a `PolygonalMesh`, this is the value returned by its `computeVolume()` method. Otherwise, the volume is 0, unless `setVolume(vol)` is used to explicitly set a non-zero volume value.
mass

A double whose default value is the product of the density and volume properties. Otherwise, if mass has been explicitly set using `setMass(mass)`, the value is the explicit mass.

density

A double whose default value is the rigid body’s density. Otherwise, if density has been explicitly set using `setDensity(density)`, the value is the explicit density, or if mass has been explicitly set using `setMass(mass)`, the value is the explicit mass divided by volume.

Note that by default, the density of a `RigidMeshComp` is simply the density setting for the rigid body, and the mass is this times the volume. However, it is possible to set either an explicit mass or a density value that will override this. (Also, explicitly setting a mass will unset any explicit density, and explicitly setting the density will unset any explicit mass.)

When the `inertiaMethod` of the rigid body is either `MASS` or `DENSITY`, then its inertia is computed from the sum of all the inertias $M_k$ of the component meshes $k$ for which `hasMass` is true. Each $M_k$ is computed by the mesh’s `createInertia(mass,massDistribution)` method, using the mass and `massDistribution` properties of its `RigidMeshComp`.

When forming the body inertia from the inertia components of individual meshes, no attempt is made to account for mesh overlap. If this is important, the meshes themselves should be modified in advance so that they do not overlap, perhaps by using the CSG primitives described in Section 2.5.7.

Instances of `RigidMeshComp` can be created directly, using constructions such as

```java
PolygonalMesh mesh;
...
initialize mesh...

RigidMeshComp mcomp = new RigidMeshComp (mesh);
```

or

```java
RigidMeshComp mcomp = new RigidMeshComp ("meshName");
mcomp.setMesh (mesh);
```

after which they can be added or removed from the `meshes` list using the methods

```java
void addMeshComp (RigidMeshComp mcomp)
void addMeshComp (RigidMeshComp mcomp, int idx)
int numMeshComps()
boolean removeMeshComp (RigidMeshComp mcomp)
boolean removeMeshComp (String name)
void clearMeshComps()
```

It is also possible to add meshes directly to the `meshes` list, using the methods

```java
RigidMeshComp addMesh (MeshBase mesh)
RigidMeshComp addMesh (MeshBase mesh, boolean hasMass, boolean collidable)
```

each of which creates a `RigidMeshComp`, adds it to the mesh list, and returns it. The second method also specifies the values of the `hasMass` and `collidable` properties (both of which are true by default).

### 3.2.10 Example: a composite rigid body

An example of constructing a rigid body from multiple meshes is defined in

`artisynth.demos.tutorial.RigidCompositeBody`

This uses three meshes to construct a rigid body whose shape resembles a dumbbell. The code, with the include files omitted, is listed below:
Figure 3.7: RigidCompositeBody loaded into ArtiSynth and run for 0.75 seconds. The ball on the right falls less because it has a lower density than the rest of the body.

```java
public class RigidCompositeBody extends RootModel {

    public void build (String[] args) {

        // create MechModel and add to RootModel
        MechModel mech = new MechModel ("mech");
        addModel (mech);

        // create the component meshes
        PolygonalMesh ball1 = MeshFactory.createIcosahedralSphere (0.8, 1);
        ball1.transform (new RigidTransform3d (1.5, 0, 0));
        PolygonalMesh ball2 = MeshFactory.createIcosahedralSphere (0.8, 1);
        ball2.transform (new RigidTransform3d (-1.5, 0, 0));
        PolygonalMesh axis = MeshFactory.createCylinder (0.2, 2.0, 12);
        axis.transform (new RigidTransform3d (0, 0, 0, 0, Math.PI/2, 0));

        // create the body and add the component meshes
        RigidBody body = new RigidBody ("body");
        body.setDensity (10);
        body.setFrameDamping (10); // add damping to the body
        body.addMesh (axis);
        RigidMeshComp bcomp1 = body.addMesh (ball1);
        RigidMeshComp bcomp2 = body.addMesh (ball2);
        mech.addRigidBody (body);

        // connect the body to a spring attached to a fixed particle
        Particle p1 = new Particle ("p1", /*mass=*/0, /*x,y,z=*/0, 0, 2);
        p1.setDynamic (false);
        mech.addParticle (p1);
        FrameMarker mkr = mech.addFrameMarkerWorld (body, new Point3d (0, 0, 0.2));
        AxialSpring spring =
            new AxialSpring (*spr*, /*k=*/150, /*d=*/0, /*restLength=*/0);
        spring.setPoints (p1, mkr);
        mech.addAxialSpring (spring);

        // set the density for ball1 to be less than the body density
        bcomp1.setDensity (8);

        // set render properties for the component, with the ball
        // meshes having different colors
    }
}
```
As in the previous examples, the `build()` method starts by creating a `MechModel` (lines 6-7). Three different meshes (two balls and an axis) are then constructed at lines 10-15, using `MeshFactory` methods (Section 2.5) and transforming each result to an appropriate position/orientation with respect to the body’s coordinate frame.

The body itself is constructed at lines 18-24. Its default density is set to 10, and its frame damping (Section 3.2.7) is also set to 10 (the previous rigid body example in Section 3.2.2 relied on spring damping to dissipate energy). The meshes are added using `addMesh()`, which allocates and returns a `RigidMeshComp`. For the ball meshes, these are saved in `bcomp1` and `bcomp2` and used later to adjust density and/or render properties.

Lines 27-34 create a simple linear spring, connected to a fixed point `p0` and a marker `mkr`. The marker is created and attached to the body by the `MechModel` method `addFrameMarkerWorld()`, which places the marker at a known position in world coordinates. The spring is created using an `AxialSpring` constructor that accepts a name, along with stiffness, damping, and rest length parameters to specify a `LinearAxialMaterial`.

At line 37, `bcomp1` is used to set the density of `ball1` to 8. Since this is less than the default body density, the inertia component of `ball1` will be lighter than that of `ball2`. Finally, render properties are set at lines 41-45. This includes setting the default face colors for the body and for each ball.

To run this example in ArtiSynth, select `All demos > tutorial > RigidCompositeBody` from the Models menu. The model should load and initially appear as in Figure 3.7. Running the model (Section 1.5.3) will cause the rigid body to fall and swing about under gravity, with the right ball (`ball1`) not falling as far because it has less density.

### 3.3 Joints and connectors

In a typical mechanical model, many of the rigid bodies are interconnected, either using spring-type components that exert binding forces on the bodies, or through joints and connectors that enforce the connection using hard constraints. This section describes the latter. While the discussion focuses on rigid bodies, joints and connectors can be used more generally with any body that implements the `ConnectableBody` interface. In particular, this allows joints to also interconnect finite element models, as described in Section 6.6.2.

#### 3.3.1 Joints and coordinate frames

Consider two rigid bodies A and B. The pose of body B with respect to body A can be described by the 6 DOF rigid transform $T_{BA}$. If A and B are unconnected, $T_{BA}$ may assume any possible value and has a full six degrees of freedom. A joint between A and B constrains the set of poses that are possible between the two bodies and reduces the degrees of freedom available to $T_{BA}$. For ease of use, the constraining action of a joint is described with respect to a pair of local coordinate frames C and D that are connected to frames A and B, respectively, by auxiliary transformations. This allows joints to be placed at locations that do not correspond directly to frames A or B.

The joint frames C and D move with respect to each other as the joint moves. The allowed joint motions therefore correspond to the allowed values of the joint transform $T_{CD}$. Although both frames typically move with their attached bodies, D is considered the base frame and C the motion frame (this is because when a joint is used to connect a single body to ground, body B is set to null and the world frame takes its place). As an example of a joint’s constraining effect, consider a hinge joint (Figure 3.8), which allows C to move with respect to D only by rotating about the z axis while the origins of C and D remain coincident. Other motions are prohibited. If we let $\theta$ describe the counter-clockwise rotation angle of C about the z axis, then $T_{CD}$ should always have the form

$$
T_{CD} = \begin{pmatrix}
\cos(\theta) & -\sin(\theta) & 0 & 0 \\
\sin(\theta) & \cos(\theta) & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{pmatrix}.
$$

(3.7)
Figure 3.8: Coordinate frames D and C for a hinge joint.

Figure 3.9: Transforms connecting joint coordinate frames C and D with rigid bodies A and B.

When a joint is attached to bodies A and B, frame C is fixed to body A and frame D is fixed to body B. Except in special cases, the joint frames C and D are not coincident with the body frames A and B. Instead, they are located relative to A and B by the transforms $T_{CA}$ and $T_{DB}$, respectively (Figure 3.9). Since $T_{CA}$ and $T_{DB}$ are both fixed, the joint constraints on $T_{CD}$ constrain the relative poses of A and B, with $T_{AB}$ determined from

$$T_{AB} = T_{DB} T_{CD} T_{CA}^{-1}. \tag{3.8}$$

(See Section A.2 for a discussion of determining transforms between related coordinate frames).

### 3.3.2 Joint coordinates, constraints, and errors

Each different joint and connector type restricts the motion between two bodies to $M$ degrees of freedom, for some $M < 6$. Sometimes, the joint also defines a set of $M$ coordinates that parameterize these $M$ DOFs. For example, the hinge joint described above is parameterized by $\theta$. Other examples are given in Section 3.4: a 2 DOF cylindrical has coordinates $z$ and $\theta$, a 3 DOF gimbal joint is parameterized by the roll-pitch-yaw angles $\theta$, $\phi$, and $\psi$, etc. When $T_{CD} = I$ (where $I$ is the identity transform), the coordinates are usually all equal to zero, and the joint is said to be in the zero state.

As explained in Section 1.2, ArtiSynth uses a full coordinate formulation for dynamic simulation. That means that instead of using joint coordinates to describe system state, it uses the combined full coordinates $q$ of all dynamic components. For example, a model consisting of a single rigid body connected to ground by a hinge joint will have 6
DOF (corresponding to the 6 DOF of the body), rather than the 1 DOF implied by the hinge joint. The DOF restrictions imposed by the joints are then enforced by a set of linearized constraint relationships

\[ G(q)u = g, \quad N(q)u \geq n \]  

(3.9)

that restrict the body velocities \( u \) computed at each simulation step, usually by solving an MLCP like (1.6). As explained in Section 1.2, the right side vectors \( g \) and \( n \) in (3.9) contain time derivative terms, which for simplicity much of the following presentation will assume to be 0.

Each joint contributes its own set of constraint equations to (3.9). Typically these take the form of bilateral, or equality, constraints

\[ G_j(q)u = 0 \]  

(3.10)

which are added to the system’s global bilateral constraint matrix \( G \). \( G \) contains \( 6 - M \) rows providing \( 6 - M \) individual constraints \( G_k \). During simulation, these give rise to \( 6 - M \) constraint forces (corresponding to \( \lambda \) in (1.8)) which enforce the constraints.

In some cases, the joint also maintains unilateral, or inequality constraints, to keep \( T_{CD} \) out of inadmissible regions. These take the form

\[ N_j(q)u \geq 0 \]  

(3.11)

and are added to the system’s global unilateral constraint matrix \( N \). They give rise to constraint forces corresponding to \( \theta \) in (1.8). A common use of unilateral constraints is to enforce range limits of the joint coordinates (Section 3.3.5), such as

\[ \theta_{\text{min}} \leq \theta \leq \theta_{\text{max}}. \]  

(3.12)

A specific unilateral constraint is added to \( N_j \) only when \( T_{CD} \) is on or within the boundary of the inadmissible region associated with that constraint. The constraint is then said to be engaged. The combined number of bilateral and engaged unilateral constraints for a particular joint should not exceed 6; otherwise, the joint would be overconstrained.

Joint coordinates, when supported for a particular joint, can be both read and set. Setting a coordinate causes the joint transform \( T_{CD} \) to change. To accommodate this, the system adjusts the poses of one or both bodies connected to the joint, along with adjacent bodies connected to them, with preference given to bodies that are not attached to “ground”. However, if this is done during simulation, and particularly if one or both of the bodies connected to the joint are moving dynamically, the results will be unpredictable and will likely conflict with the simulation.

Joint coordinates are also often exported as properties. For example, the \texttt{HingeJoint} class (Section 3.4) exports its \( \theta \) coordinate as the property \texttt{theta}, which can be accessed in the GUI, or via the accessor methods

```java
double getTheta() // get theta in degrees
void setTheta (double deg) // set theta in degrees
```

Since joint constraints are generally nonlinear, their linearized enforcement at the velocity level by (3.9) will usually produce small errors as the simulation proceeds. These errors are reduced using a position correction step described in Section 4.8.1 and [9]. Errors can also be caused by joint compliance (Section 3.3.8). Both effects mean that the joint transform \( T_{CD} \) may deviate from the allowed values dictated by the joint type. In ArtiSynth, this is accounted for by introducing an additional constraint frame \( G \) between \( D \) and \( C \) (Figure 3.10). \( G \) is computed to be the nearest frame to \( C \) that lies exactly in the joint constraint space. \( T_{GD} \) is therefore a valid joint transform, \( T_{GC} \) accommodates the error, and the whole joint transform is given by the composition

\[ T_{CD} = T_{GD} T_{CG}. \]  

(3.13)

If there is no compliance or joint error, then frames \( G \) and \( C \) are identical, \( T_{CG} = I \), and \( T_{CD} = T_{GD} \). Because \( T_{CG} \) describes the joint error, we sometimes refer to it as \( T_{\text{err}} = T_{CG} \).

### 3.3.3 Creating Joints

Joint and connector components in ArtiSynth are both derived from the superclass \texttt{BodyConnector}, with joints being further derived from \texttt{JointBase}, which provides support for coordinates. Some of the commonly used joints and connectors are described in Section 3.4.

An application creates a joint by constructing it and adding it to a \texttt{MechModel}. Many joints have constructors of the form

```java
XXXJoint (bodyA, bodyB, TDW)
```
Figure 3.10: 2D schematic showing the joint frames D and C, along with the intermediate frame G that accounts for numeric error and complaint motion.

which specifies the bodies A and B which the joint connects, along with the transform $T_{DW}$ giving the pose of the joint base frame D in world coordinates. The constructor then assumes that the joint is in the zero state, so that C and D are the same and $T_{CD} = I$ and $T_{CW} = T_{DW}$, and then computes $T_{CA}$ and $T_{DB}$ from

$$T_{CA} = T_{AW}^{-1} T_{CW}$$

$$T_{DB} = T_{BW}^{-1} T_{DW},$$

where $T_{AW}$ and $T_{BW}$ are the current poses of A and B.

After the joint is created, it should be added to the system’s `MechModel` using `addBodyConnector()`, as shown in the following code fragment:

```java
MechModel mech;
RigidBody bodyA, bodyB;
RigidTransform3d TDW;
...
initialize mech, bodyA, bodyB, and TDW ...

HingeJoint joint = new HingeJoint (bodyA, bodyB, TDW);
mech.addBodyConnector (joint);
```

It is also possible to create a joint using its default constructor and attach it to the bodies afterward, using the method `setBodies(bodyA, bodyB, TDW)`, as in the following:

```java
HingeJoint joint = new HingeJoint ();
joint.setBodies (bodyA, bodyB, TDW);
mech.addBodyConnector (joint);
```

One reason for doing this is that it allows the joint transform $T_{CD}$ to be modified (by setting coordinate values) before `set Bodies()` is called; this is discussed further in Section 3.3.4.

Joints usually offer a number of other constructors that let its world location and body relationships to be specified in different ways. These may include:

```java
XXXJoint (bodyA, TCA, bodyB, TDB)
XXXJoint (bodyA, bodyB, TCW, TDW)
```

The first, which is restricted to rigid bodies, allows the application to explicitly specify transforms $T_{CA}$ and $T_{DB}$ connecting frames C and D to the body frames A and B, and is useful when $T_{CA}$ and $T_{DB}$ are explicitly known, or the
initial value of $T_{CD}$ is not the identity. Likewise, the second constructor allows $T_{CW}$ and $T_{DW}$ to be explicitly specified, with $T_{CD} \neq I$ if $T_{CW} \neq T_{DW}$. For instance, suppose $T_{CD}$ and $T_{DW}$ are both known. Then we can use the relationship

$$T_{CW} = T_{DW} T_{CD}$$

(3.16)

to create the joint as in the following code fragment:

```java
MechModel mech;
RigidBody bodyA, bodyB;
RigidTransform3d TDW, TCD;
...
initialize mech, bodyA, bodyB, TDW, and TCD ...

// compute TCW:
RigidTransform3d TCW = new RigidTransform3d();
TCW.mul (TDW, TCD);
HingeJoint joint = new HingeJoint (bodyA, bodyB, TCW, TDW);
mech.addBodyConnector (joint);
```

As an alternative to specifying $T_{DW}$ or its equivalents, some joint types provide constructors that let the application locate specific joint features. These may be easier to use in some cases. For instance, `HingeJoint` provides a constructor

```java
HingeJoint (bodyA, bodyB, originD, zaxis)
```

that specifies origin of D and its z axis (which is the rotation axis), with the remaining orientation of D aligned as closely as possible with the world. `SphericalJoint` provides a constructor

```java
SphericalJoint (bodyA, bodyB, originD)
```

that specifies origin of D and aligns its orientation with the world. Users should consult the source code or API documentation for specific joints to see what special constructors may be available.

Finally, it is possible to use joints to connect a single body to ground (by convention, this is the A body). Most joints provide a constructor of the form

```java
XXXJoint (bodyA, TDW)
```

which allows this to be done explicitly. Alternatively, most joint constructors which supply body B will allow this to be specified as null, so that body A will be connected to ground by default.

### 3.3.4 Working with coordinates

As mentioned in Section 3.3.2, some joints support coordinates that parameterize the valid motions within the joint transform $T_{CD}$. All such joints are subclasses of `JointBase`, which provides some generic methods for querying and setting coordinate values (`JointBase` is in turn a subclass of `BodyConnector`).

The number of coordinates is returned by the method `numCoordinates()`; if this returns 0, then coordinates are not supported. Each coordinate has an index in the range $0 \ldots M - 1$, where $M$ is the number of coordinates. Coordinate values can be queried or set using the following methods:

```java
getCoordinate (int idx) // get coordinate value with index idx
getCoordinates (VectorNd coords) // get all coordinates values
setCoordinate (int idx, double value) // set coordinate value with index idx
setCoordinates (VectorNd values) // set all coordinates values
```

Specific joint types usually also provide names for their joint coordinates, along with integer constants describing their indices and methods for accessing their values. For example, `CylindricalJoint` supports two coordinates, $z$ and $\theta$, along with the following:

```java
// coordinate indices
static final int Z_IDX = 0;
static final int THETA_IDX = 1;
```
// set/get z value and range
doctor getZ()  
void setZ (double z)

// set/get theta value and range in degrees
double getTheta()  
void setTheta (double theta)

The coordinate values are also exported as the properties z and theta, allowing them to be set in the GUI. For convenience, particularly in GUI applications, the properties and methods for controlling specific angular coordinates generally use degrees instead of radians.

As discussed in Section 3.3.2, unlike in some multibody simulation systems (such as OpenSim), joint coordinates are not fundamental quantities that describe system state. As such, then, coordinates can usually only be set in specific circumstances that avoid simulation conflicts. In general, when joint coordinates are set, the system adjusts the poses of one or both bodies connected to this joint, along with adjacent bodies connected to them, with preference given to bodies that are not attached to “ground”. However, if this is done during simulation, and particularly if one or both of the bodies connected to the joint are moving dynamically, the results will be unpredictable and will likely conflict with the simulation.

If a joint has been created with its default constructor and not yet attached to any bodies, then setting joint values will simply set the joint transform \( T_{CD} \). This can be useful in situations where one needs to initialize a joint’s \( T_{CD} \) to a non-identity value corresponding to a particular set of joint coordinates:

```java
RigidTransform3d TDW; // known location for D frame
doctor z, theta; // desired initial coordinate values
...
CylindricalJoint joint = new CylindricalJoint();
joint.setZ (z);
joint.setTheta (thetaDeg);
joint.setBodies (bodyA, bodyB, TDW);
```

This can also be done in vector form:

```java
RigidTransform3d TDW; // known location for D frame
VectorNd coordValues; // desired initial coordinate values
...
CylindricalJoint joint = new CylindricalJoint();
joint.setCoordinates (coordValues);
joint.setBodies (bodyA, bodyB, TDW);
```

In either of these cases, `setBodies()` will not use \( T_{CD} = I \) but instead use the value determined by the initial coordinate values.

To determine the \( T_{CD} \) corresponding to a particular set of coordinates, one may use the method

```java
void coordinatesToTCD (RigidTransform3d TCD, VectorNd coords)
```

In some cases, within a model’s `build()` method, one may wish to set initial coordinates after a joint has been attached to its bodies, in order to move those bodies (along with the bodies attached to them) into an initial configuration without having to explicitly calculate the poses from the joint coordinates. As mentioned above, the system will make a decision about which attached bodies are most “free” and adjust their poses accordingly. This is done in the example of the next section.

### 3.3.5 Coordinate limits and locking

It is possible to set limits on a joint coordinate’s range, and also to lock a coordinate in place at its current value.

When a joint coordinate hits either an upper or lower range limit, a unilateral constraint is invoked to prevent it from violating the limit, and remains engaged until the joint moves away from the limit. Each range constraint that is engaged reduces the number of joint DOFs by one.

By default, joint range limits are usually disabled (i.e., they are set to \((-\infty, \infty)\)). They can be queried and set, for a given joint with index `idx`, using the methods:
where range limits for angular coordinates are specified in radians. For convenience, the following methods are also provided which use degrees instead of radians for angular coordinates:

```java
DoubleInterval getCoordinateRangeDeg (int idx)
double getMinCoordinateDeg (int idx)
double getMaxCoordinateDeg (int idx)
void setCoordinateRangeDeg (idx, DoubleInterval rng)
```

Range checking can be disabled by setting the range to \((-\infty, \infty)\), or by specifying `rng` as `null`, which implicitly does the same thing. Examples of this in

Ranges for angular coordinates are not limited to \(\pm 180^\circ\) but can instead be set to larger values; the joint will continue to wrap until the limit is reached.

Joint coordinates can also be `locked`, so that they hold their current value and don’t move. A joint is locked using a bilateral constraint that prevents motion in either direction and reduces the joint’s DOF count by one. The following methods are available for querying or setting a coordinate’s locked status:

```java
boolean isLocked (int idx)
void setLocked (int idx, boolean locked)
```

As with coordinate values, specific joint types usually provide methods for controlling the ranges and locking status of individual coordinates, with ranges for angular coordinates specified in degrees instead of radians. For example, `CylindricalJoint` supplies the methods

```java
// set/get z range
DoubleInterval getZRange()
void setZRange (double min, double max)

// control z locking
boolean isZLocked()
void setZLocked (boolean locked)

// set/get theta range in degrees
DoubleInterval getThetaRange()
void setThetaRange (double min, double max)
void setThetaRange (DoubleInterval rng)

// control theta locking
boolean isThetaLocked()
void setThetaLocked (boolean locked)
```

The range and locking information is also exported as the properties `zRange`, `thetaRange`, `zLocked`, and `thetaLocked`, allowing them to be set in the GUI.

### 3.3.6 Example: a simple hinge joint

A simple model showing two rigid bodies connected by a joint is defined in

```
artisynth.demos.tutorial.RigidBodyJoint
```

The build method for this model is given below:
public void build (String[] args) {

    // create MechModel and add to RootModel
    mech = new MechModel ("mech");
    mech.setGravity (0, 0, -98);
    mech setFrameDamping (1.0);
    mech.setRotaryDamping (4.0);
    addModel (mech);

    PolygonalMesh mesh; // bodies will be defined using a mesh

    // create first body and set its pose
    mesh = MeshFactory.createRoundedBox (lenx1, leny1, lenz1, /*nslices=*/8);
    RigidTransform3d TMB =
            new RigidTransform3d (0, 0, 0, /*axisAng=*/1, 1, 1, 2*Math.PI/3);
    mesh.transform (TMB);
    bodyB = RigidBody.createFromMesh ("bodyB", mesh, /*density=*/0.2, 1.0);
    bodyB.setPose (new RigidTransform3d (0, 0, 1.5*lenx1, 1, 0, 0, Math.PI/2));
    bodyB.setDynamic (false);

    // create second body and set its pose
    leny2/2, lenx2, /*nslices=*/16, /*nsegs=*/1, /*flatBottom=*/false);
    mesh.transform (TMB);
    bodyA = RigidBody.createFromMesh ("bodyA", mesh, 0.2, 1.0);
    bodyA.setPose (new RigidTransform3d (
            (lenx1+lenx2)/2, 0, 1.5*lenx1, 1, 0, 0, Math.PI/2));

    // create the joint
    RigidTransform3d TDW =
            new RigidTransform3d (lenx1/2, 0, 1.5*lenx1, 1, 0, 0, Math.PI/2);
    HingeJoint joint = new HingeJoint (bodyA, bodyB, TDW);

    // add components to the mech model
    mech.addRigidBody (bodyB);
    mech.addRigidBody (bodyA);
    mech.addBodyConnector (joint);

    joint.setTheta (35); // set joint position

    // set render properties for components

A MechModel is created as usual at line 4. However, in this example, we also set some parameters for it: setGravity() is used to set the gravity acceleration vector to \((0, 0, -9.8)\) instead of the default value of \((0, 0, -9)\), and the frameDamping and rotaryDamping properties (Section 3.2.7) are set to provide appropriate damping.

Each of the two rigid bodies are created from a mesh and a density. The meshes themselves are created using the factory methods MeshFactory.createRoundedBox() and MeshFactory.createRoundedCylinder() (lines 13 and 22), and then RigidBody.createFromMesh() is used to turn these into rigid bodies with a density of 0.2 (lines 17 and 25). The pose of the two bodies is set using RigidTransform3d objects created with x, y, z translation and axis-angle orientation values (lines 18 and 26).

The hinge joint is implemented using HingeJoint, which is constructed at line 32 with the joint coordinate frame D being located in world coordinates by TDW as described in Section 3.3.3.

Once the joint is created and added to the MechModel, the method setTheta() is used to explicitly set the joint parameter to 35 degrees. The joint transform \(T_{CD}\) is then set appropriately and bodyA is moved to accommodate this (bodyA being chosen since it is the most free to move).

Finally, joint rendering properties are set starting at line 42. We render the joint as a cylindrical shaft about the rotation axis, using its shaftLength and shaftRadius properties. Joint rendering is discussed in more detail in Section 3.3.10.

To run this example in ArtiSynth, select All demos > tutorial > RigidBodyJoint from the Models menu. The model should load and initially appear as in Figure 3.11. Running the model (Section 1.5.3) will cause bodyA to fall and swing under gravity.

### 3.3.7 Constraint forces

During each simulation solve step, the joint velocity constraints described by (3.10) and (3.11) are enforced by bilateral and unilateral constraint forces \(f_g\) and \(f_n\):

\[
\begin{align*}
  f_g &= G_J^T \lambda_J, \\
  f_n &= N_J^T \theta_J.
\end{align*}
\]  

(3.17)

Here, \(f_g\) and \(f_n\) are spatial forces (or wrenches, Section A.5) acting in the joint coordinate frame C, and \(\lambda_J\) and \(\theta_J\) are the Lagrange multipliers computed as part of the mechanical system solve (see (1.6) and (1.8)). The sizes of \(\lambda_J\) and \(\theta_J\) equal the number of bilateral and engaged unilateral constraints in the joint; these numbers can be queried for a particular joint using the methods numBilateralConstraints() and numEngagedUnilateralConstraints(). (The number of engaged unilateral constraints may be less than the total number of unilateral constraints; the latter may be queried with numUnilateralConstraints(), while the total number of constraints is returned by numConstraints().

Applications may sometimes need to query the current constraint force values, typically from within a controller or monitor (Section 5.3). The Lagrange multipliers themselves may be obtained with

```java
void getBilateralForces (VectorNd lam)
void getUnilateralForces (VectorNd the)
```

which load the multipliers into lam or the and set their sizes to the number of bilateral or engaged unilateral constraints. Alternatively, one can retrieve the individual multiplier for the constraint indexed by idx using

```java
double getConstraintForce (int idx);
```

Typically, it is more useful to find the spatial constraint forces \(f_g\) and \(f_n\), which can be obtained with respect to frame C:

```java
// place the forces in the wrench f
void getBilateralForcesInC (Wrench f)
void getUnilateralForcesInC (Wrench f)

// convenience methods that allocate the wrench and return it
Wrench getBilateralForcesInC();
Wrench getUnilateralForcesInC();
```
If the attached bodies A and B are rigid bodies, it is also possible to obtain the constraint wrenches experienced by those bodies:

```cpp
// place the forces in the wrench f
void getBilateralForcesInA (Wrench f)
void getUnilateralForcesInA (Wrench f)
void getBilateralForcesInB (Wrench f)
void getUnilateralForcesInB (Wrench f)

// convenience methods that allocate the wrench and return it
Wrench getBilateralForcesInA ();
Wrench getUnilateralForcesInA ();
Wrench getBilateralForcesInB ();
Wrench getUnilateralForcesInB ();
```

Constraint wrenches obtained for bodies A or B are given in world coordinates, which is consistent with the forces reported by rigid bodies via their `getForce()` method. To orient the forces into body coordinates, one may use the inverse of the rotation matrix \( R \) of the body’s pose. For example:

```cpp
RigidBody bodyA;

// ... body A initialized, etc. ...
Wrench force = joint.getBilateralForceInA ();
force.inverseTransform (bodyA.getPose ().R);
```

### 3.3.8 Compliance and regularization

By default, the constraints used to implement joints and couplings are treated as hard, so that the system tries to respect the constraint conditions (3.9) as exactly as possible as the simulation proceeds. Sometimes, however, it is desirable to introduce some “softness” into the constraints, whereby constraint forces are determined as a linear function of their distance from the constraint. Adding compliance also allows an application to regularize a system of joint constraints that would otherwise be overconstrained, as illustrated in Section 3.3.9.

To describe compliance precisely, consider the bilateral constraint portion of the MLCP in (1.6), which solves for the updated system velocities \( u_{k+1} \) at each time step:

\[
\begin{pmatrix}
\hat{M}^k & -G^T \\
G & 0 \\
\end{pmatrix}
\begin{pmatrix}
u_{k+1} \\
\hat{\lambda} \\
\end{pmatrix}
= \begin{pmatrix}
Mu_k - h\hat{f}^k \\
0 \\
\end{pmatrix}.
\] (3.18)

Here \( G \) is the system’s bilateral constraint matrix, \( \hat{\lambda} \) denotes the constraint impulses (from which the constraint forces \( \lambda \) can be determined by \( \lambda = \hat{\lambda} / h \)), and for simplicity we have assumed that \( G \) is constant and so the \( g \) term on the lower right side is 0.

Solving (3.18) results in constraint forces that satisfy \( Gu_{k+1} = 0 \) precisely, corresponding to hard constraints. To implement soft constraints, start by defining a function \( \phi (q) \) that defines the distances from each constraint, where \( q \) is the vector of system positions; these distances are the local translational and rotational deviations from each constraint’s correct position and are discussed in more detail in Section 4.8.1. Then assume that the constraint forces are a linear function of these distances:

\[
\lambda = -C^{-1}\phi (q),
\] (3.19)

where \( C \) is a diagonal compliance matrix that is equivalent to an inverse stiffness matrix. We also note that \( \phi \) will be time varying, and that we can approximate its change between time steps as

\[
\phi^{k+1} \approx \phi^k + h\phi^{k+1}, \quad \text{with} \quad \phi^{k+1} = Gu^{k+1}.
\] (3.20)

Next, assume that in using (3.19) to determine \( \lambda \) for a particular time step, we use the average value of \( \phi \) over the step, represented by \( \bar{\phi} = (\phi^{k+1} + \phi^k) / 2 \). Substituting this and (3.20) into (3.19), multiplying by \( C \), and rearranging yields:

\[
Gu^{k+1} + \frac{2C}{h}\bar{\phi} = -\frac{2}{h}\phi^k.
\] (3.21)
Then noting that $\tilde{\lambda} = h\lambda$, we obtain a revised form of (3.18),

$$
\begin{pmatrix}
\dot{\mathbf{M}}^k & -\mathbf{G}^T \\
\mathbf{G} & 2\mathbf{C}/h^2
\end{pmatrix}
\begin{pmatrix}
\mathbf{u}^{k+1} \\
\tilde{\lambda}
\end{pmatrix}
= 
\begin{pmatrix}
(M\mathbf{u}^k - h\tilde{\lambda}) \\
-2\phi^k/h
\end{pmatrix},
$$

in the which the zeros in the matrix and right hand side have been replaced by compliance terms. The resulting constraint behavior is different from that of (3.18) in two important ways:

1. The joint now allows 6 DOF, with motion along the constrained directions limited by restoring spring constants given by the reciprocals of the diagonal entries of $\mathbf{C}$.

2. If $\mathbf{C}$ has no zero diagonal entries, then the system (3.22) is regularized by the $2\mathbf{C}/h^2$ term in the lower right matrix block. This means that the matrix is always non-singular, even if $\mathbf{G}$ is rank deficient, and so compliance offers a way to handle overconstrained models, as discussed further in Section 3.3.9.

Unilateral constraints can be regularized using the same approach, with a distance function defined such that $\phi(q) \leq 0$.

The reason for specifying soft constraints using compliance instead of stiffness is that by setting $\mathbf{C} = 0$ we can easily handle the case of infinite stiffness where the constraints are strictly enforced. The ArtiSynth compliance implementation uses a slightly more complex version of (3.22) that accounts for non-constant $\mathbf{G}$ and also allows for a damping term $-D\dot{\phi}$, where $\mathbf{D}$ is again a diagonal matrix. For more details, see [7] and [17].

When using compliance, damping is often needed for stability, and, in the case of unilateral constraints, to prevent “bouncing”. A good choice for damping is usually critical damping, which is discussed further below.

Any joint which is a subclass of `BodyConnector` allows individual compliance values $C_i$ and damping values $D_i$ to be set for each of the joint’s $i$ constraints. These values comprise the diagonal entries in the compliance and damping matrices $\mathbf{C}$ and $\mathbf{D}$, and can be queried and set using the methods

```cpp
VectorNd getCompliance() 
void setCompliance(VectorNd compliance)
VectorNd getDamping()
void setCompliance(VectorNd damping)
```

The vectors supplied to the above set methods contain the requested compliance or damping values. If their size $n$ is less than `numConstraints()`, then compliance or damping will be set for the first $n$ constraints. Damping for a specific constraint only has an effect if the compliance for that constraint is nonzero.

What compliance and damping values should be specified? Compliance is usually relatively easy to figure out. Each of the joint’s individual constraints $i$ corresponds to a row in its bilateral constraint matrix $\mathbf{G}_i$ or unilateral constraint matrix $\mathbf{N}_i$, and represents a specific 6 DOF direction along which the spatial velocity $\dot{\mathbf{v}}_{CD}$ (of frame C with respect to D) is restricted (more details on this are given in Section 4.8.1). Each of these constraint directions is usually predominantly linear or rotational; specific descriptions for the constraints of different joints are provided in Section 3.4. To determine compliance for a constraint $i$, estimate the typical force $f$ likely to act along its direction, decide how much displacement $\delta q$ (translational or rotational) along that constraint is desirable, and then set the compliance $C_i$ to the associated inverse stiffness:

$$C_i = \frac{\delta q}{f}.$$  
(3.23)

Once $C_i$ is determined, the damping $D_k$ can be estimated based on the desired damping ratio $\zeta$, using the formula

$$D_k = 2\zeta \sqrt{M/C_k},$$

(3.24)

where $M$ is total mass of the bodies attached to the joint. Typically, the desired damping will be close to critical damping, for which $\zeta = 1$.

Constraints associated with linear motion will typically require different compliance values from those associated with rotation. To make this process easier, joint components allow the setting of collective compliance values for their linear and rotary constraints, using the methods

```cpp
void setLinearCompliance (double c)
double getLinearCompliance ()
void setRotaryCompliance (double c)
double getRotaryCompliance ()
```
The set() methods will set a uniform compliance for all linear or rotary constraints, except for unilateral constraints associated with coordinate limits. At the same time, they will also set an automatically computed critical damping value. Likewise, the get() methods query these linear or rotary constraints for uniform compliance values (with the corresponding critical damping), and return either that value, or -1 if it does not exist.

Most of the demonstration models for the joints described in Section 3.4 allow these linear and rotary compliance settings to be adjusted interactively using a control panel, enabling users to experimentally gain a feel for their behavior. To determine programmatically whether a particular constraint is linear or rotary, one can use the joint method

```java
VectorNi getConstraintFlags()
```

which returns a vector of information flags for all its constraints. Linear and rotary constraints are indicated by the flags LINEAR and ROTARY, defined in RigidBodyConstraint.

### 3.3.9 Example: an overconstrained linkage

Situations may occasionally arise in which a model is overconstrained, which means that the rows of the bilateral constraint matrix $G$ in (3.9) are not all linearly dependent, or in other words, $G$ does not have full row rank. At present, the ArtiSynth solver has difficulty handling overconstrained models, but these situations can often be handled by adding a small amount of compliance to the constraints. (Overconstraining is not a problem with unilateral constraints $N$, because of the way they are handled by the solver.)

One possible symptom of an overconstrained system is an error message in the application’s terminal output, such as

```
Pardiso: num perturbed pivots=12
```

Overconstraining frequently occurs in closed-chain linkages, involving loops in which a jointed sequence of links is connected back on itself. Depending on how the constraints are configured and how redundant they are, the system may still be able to move. A classical example is the four-bar linkage, a common version of which consists of four links, or “bars”, arranged as a parallelogram and connected by hinge joints at the corners. One link is usually connected to ground, and so the remaining three links together have 18 DOF, while the four hinge joints together remove 20 DOF, overconstraining the system. However, the constraints are redundant in such a way that the linkage still actually has 1 DOF.

![Figure 3.12: FourBarLinkage model, several steps into the simulation.](image)

To model a four-bar in ArtiSynth presently requires adding compliance to the hinge joints. An example of this is defined by the demo program

`artisynth.demos.tutorial.FourBarLinkage`
shown in Figure 3.12. The code for the build() method and a couple of supporting methods is given below:

```java
/**
 * Create a link with a length of 1.0, width of 0.25, and specified depth
 * and add it to the mech model. The parameters x, z, and deg specify the
 * link's position and orientation (in degrees) in the x-z plane.
 */
protected RigidBody createLink (MechModel mech, String name,
double depth, double x, double z, double deg) {
    int nslices = 20; // num slices on the rounded mesh ends
    PolygonalMesh mesh = MeshFactory.createRoundedBox (1.0, 0.25, depth, nslices);
    RigidBody body = RigidBody.createFromMesh (name, mesh, /* density=*/1000.0, /* scale=*/1.0);
    body.setPose (new RigidTransform3d (x, 0, z, 0, Math.toRadians (deg), 0));
    mech.addRigidBody (body);
    return body;
}

/**
 * Create a hinge joint connecting one end of link0 with the other end of
 * link1, and add it to the mech model.
 */
protected HingeJoint createJoint (MechModel mech, String name,
RigidBody link0, RigidBody link1) {
    // easier to locate the link using TCA and TDB since we know where frames
    // C and D are with respect the link0 and link1
    RigidTransform3d TCA = new RigidTransform3d (0, 0, 0.5, 0, 0, Math.PI/2);
    RigidTransform3d TDB = new RigidTransform3d (0, 0, -0.5, 0, 0, Math.PI/2);
    HingeJoint joint = new HingeJoint (link0, TCA, link1, TDB);
    joint.setName (name);
    mech.addBodyConnector (joint);
    // set joint render properties
    joint.setAxisLength (0.4);
    RenderProps.setLineRadius (joint, 0.03);
    return joint;
}

public void build (String[] args) {
    // create a mech model and set rigid body damping parameters
    MechModel mech = new MechModel("mech");
    addModel (mech);
    mech.setFrameDamping (1.0);
    mech.setRotaryDamping (4.0);

    // create four 'bars' from which to construct the linkage
    RigidBody[] bars = new RigidBody[4];
bars[0] = createLink (mech, "link0", 0.2, -0.5, 0.0, 0);
bars[1] = createLink (mech, "link1", 0.3, 0.0, 0.5, 90);
bars[2] = createLink (mech, "link2", 0.2, 0.5, 0.0, 180);
bars[3] = createLink (mech, "link3", 0.3, 0.0, -0.5, 270);
    // ground the left bar
    bars[0].setDynamic (false);

    // connect the bars using four hinge joints
    HingeJoint[] joints = new HingeJoint[4];
joints[0] = createJoint (mech, "joint0", bars[0], bars[1]);
joints[1] = createJoint (mech, "joint1", bars[1], bars[2]);
joints[2] = createJoint (mech, "joint2", bars[2], bars[3]);
joints[3] = createJoint (mech, "joint3", bars[3], bars[0]);

    // Set uniform compliance and damping for all bilateral constraints,
    // which are the first 5 constraints of each joint
    VectorNd compliance = new VectorNd(5);
```
Two helper methods are used to construct the model: `createLink()` (lines 6-17), and `createJoint()` (lines 23-36). `createLink()` makes the individual rigid bodies used to build the linkage: a mesh is produced defining the body’s shape (a box with rounded ends), and then passed to the `RigidBody createFromMesh()` method which creates the body and sets its inertia according to a specified density. The body’s pose is then set so as to center it at \((x, 0, z)\) while rotating it about the \(y\) axis by the angle \(\text{deg}\) (in degrees). The completed body is then added to the `MechModel mech` and returned.

The second helper method, `createJoint()`, connects two rigid bodies (`link0` and `link1`) together using a `HingeJoint`. Because we know the location of the joint in body-relative coordinates, it is easier to create the joint using the transforms \(T_{CA}\) and \(T_{DB}\) instead of \(T_{DW}: T_{CA}\) locates the joint at the top end of `link0`, at \((0, 0, 0.5)\), with the \(z\) axis parallel to the body’s \(y\) axis, while \(T_{DB}\) similarly locates the joint at the bottom of `link1`. After the joint is created and added to the `MechModel`, its render properties are set so that its axis drawn as a blue cylinder.

The `build()` method itself begins by creating a `MechModel` and setting damping parameters for the rigid bodies (lines 40-43). Next, `createLink()` is used to create and store the four links (lines 46-50), and the left bar is attached to ground by making it non-dynamic (line 52). The links are then connected together using joints created by `createJoint()` (lines 55-59). Finally, uniform compliance and damping values are set for each of the joint’s bilateral constraints, using the `setCompliance()` and `setDamping()` methods (lines 63-72). Values are set for the first five constraints, since for a `HingeJoint` these are the bilateral constraints. The compliance value of \(C = 10^{-6}\) was found experimentally to be low enough so as to not cause noticeable deflections in the joints. Given \(C\) and an average mass of around \(M = 150\) for each link pair, \((3.24)\) suggests the damping factor of \(D = 25000\). Note that for this example, very similar settings could be achieved by simply calling

```java
for (int i=0; i< joints.length; i++) {
    joints[i].setLinearCompliance (0.000001);
    joints[i].setRotaryCompliance (0.000001);
}
```

In principle, we only need to set compliance for the constraints that are redundant, but it can sometimes be difficult to determine exactly which these are. Also, different values are often needed for linear and rotary constraints; that is not necessary here because the links have unit length and so the linear and rotary units have similar scales.

### 3.3.10 Rendering joints

Most joints provide a means to render themselves in order to provide a graphical representation of their position and configuration. Control over this is achieved by setting various properties in the joint component, including both specialized properties and the standard render properties (Section 4.3) used by all renderable components.

All joints which are subclasses of `JointBase` support rendering of both their `C` and `D` coordinate frames, through the properties `drawFrameC`, `drawFrameD`, and `axisLength`. The first two properties are of the type `Renderer.AxisDrawStyle` (described in detail in Section 3.2.8), and can be set to `LINE` or `ARROW` to enable the coordinate axes to be drawn either as lines or solid arrows. The `axisLength` property has type `double` and specifies the length with which the axes are drawn. As with all properties, these properties can be set either in the GUI, or in code using accessor methods supplied by the joint:

```java
void setAxisLength (double l)
double getAxisLength ()

void setDrawFrameC (AxisDrawStyle style)
(AxisDrawStyle getDrawFrameC())
```
Another pair of properties used by several joints is `shaftLength` and `shaftRadius`, which specify the length and radius used to draw shaft or axis structures associated with the joint. These are rendered as solid cylinders, using the color indicated by the `faceColor` rendering property. The default value of both properties is 0; if `shaftLength` is 0, then the structures are not drawn, while if `shaftRadius` is 0, a default value proportional to `shaftLength` is used. For example, to enable rendering of a blue shaft along the rotation axis of a hinge joint, one may use the code fragment

```java
HingeJoint joint;
...
joint.setShaftLength (0.5); // set shaft dimensions
joint.setShaftRadius (0.05);
RenderProps.setFaceColor (joint, Color.BLUE); // set the color
```

As another example, to enable rendering of a green ball about the center of a spherical joint, one may use the fragment

```java
SphericalJoint joint;
...
joint.setJointRadius (0.02); // set the ball size
RenderProps.setFaceColor (joint, Color.GREEN); // set the color
```

Specific joints may define additional properties to control how they are rendered.

### 3.4 Joint components

ArtiSynth supplies a number of basic joints and connectors in the package `artisynth.core.mechmodels`, the most common of which are described here.

Many of the descriptions are associated with a demonstration model, named `XXXJointDemo`, where `XXX` is the joint type. These demos are located in the package `artisynth.demos.mech`, and can be loaded by selecting `All demos > mech > XXXJointDemo` from the `Models` menu. When run, they can be interactively controlled, using either the pull tool (see the section “Pull Manipulation” in the `ArtiSynth User Interface Guide`), or the interactive control panel. The control panel allows the adjustment of coordinate values and ranges (if supported), some of the render properties, and the different compliance and damping properties (Section 3.3.8). One can inspect the source code for each demo in its `.java` file located in the folder `<ARTISYNTH_HOME>/src/artisynth/demos/mech`.

#### 3.4.1 Hinge Joint

The **HingeJoint** (Figure 3.13) is a 1 DOF joint that constrains motion between frames C and D to a simple rotation about the z axis of D. It implements six constraints and one coordinate θ (Table 3.1), to which the joint transform $T_{CD}$ is...
related by

\[
T_{CD} = \begin{pmatrix}
\cos(\theta) & -\sin(\theta) & 0 & 0 \\
\sin(\theta) & \cos(\theta) & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{pmatrix}.
\]

The value and ranges for \( \theta \) are exported by the properties \texttt{theta} and \texttt{thetaRange}, and the \( \theta \) coordinate index is defined by the constant \texttt{THETA_IDX}. For rendering, the properties \texttt{shaftLength} and \texttt{shaftRadius} control the size of a shaft drawn about the rotation axis, using the \texttt{faceColor} rendering property. A demo is provided by \texttt{artisynth.demos.mech.HingeJointDemo}.

In addition to the standard constructors described in Section 3.3.3,

\begin{verbatim}
HingeJoint (bodyA, bodyB, originD, zaxis)
\end{verbatim}

creates a hinge joint with a specified origin and \( z \) axis direction for frame D (in world coordinates), and frames C and D coincident.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along ( x )</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along ( y )</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts translation along ( z )</td>
</tr>
<tr>
<td>3</td>
<td>bilateral</td>
<td>restricts rotation about ( x )</td>
</tr>
<tr>
<td>4</td>
<td>bilateral</td>
<td>restricts rotation about ( y )</td>
</tr>
<tr>
<td>5</td>
<td>unilateral</td>
<td>enforces limits on ( \theta )</td>
</tr>
</tbody>
</table>

\begin{verbatim}
0 \( \theta \) counter-clockwise rotation of \( C \) about the \( z \) axis
\end{verbatim}

Table 3.1: Constraints (top) and coordinates (bottom) for the hinge joint.

### 3.4.2 Slider Joint

![Slider Joint](image)

Figure 3.14: Coordinate frames (left) and demo model (right) for the slider joint.

The \texttt{SliderJoint} (Figure 3.14) is a 1 DOF joint that constrains motion between frames C and D to a simple translation along the \( z \) axis of D. It implements six constraints and one coordinate \( z \) (Table 3.2), to which the joint transform \( T_{CD} \) is related by

\[
T_{CD} = \begin{pmatrix}
1 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 \\
0 & 0 & 1 & z \\
0 & 0 & 0 & 1
\end{pmatrix}.
\]

The value and ranges for \( z \) are exported by the properties \texttt{z} and \texttt{zRange}, and the \( z \) coordinate index is defined by the constant \texttt{Z_IDX}. For rendering, the properties \texttt{shaftLength} and \texttt{shaftRadius} control the size of a shaft drawn about the sliding axis, using the \texttt{faceColor} rendering property. A demo is provided by \texttt{artisynth.demos.mech.SliderJointDemo}.

In addition to the standard constructors described in Section 3.3.3,

\begin{verbatim}
SliderJoint (bodyA, bodyB, originD, zaxis)
\end{verbatim}
creates a slider joint with a specified origin and z axis direction for frame D (in world coordinates), and frames C and D coincident.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along x</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along y</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts rotation about x</td>
</tr>
<tr>
<td>3</td>
<td>bilateral</td>
<td>restricts rotation about y</td>
</tr>
<tr>
<td>4</td>
<td>bilateral</td>
<td>restricts rotation about z</td>
</tr>
<tr>
<td>5</td>
<td>unilateral</td>
<td>enforces limits on the z coordinate</td>
</tr>
<tr>
<td>0</td>
<td>z</td>
<td>translation of C along the z axis</td>
</tr>
</tbody>
</table>

Table 3.2: Constraints (top) and coordinates (bottom) for the slider joint.

3.4.3 Cylindrical Joint

![Cylindrical Joint Diagram](image)

The CylindricalJoint (Figure 3.15) is a 2 DOF joint that constrains motion between frames C and D to translation and rotation along and about the z axis of D. It implements six constraints and two coordinates z and θ (Table 3.3), to which the joint transform $T_{CD}$ is related by

$$
T_{CD} = \begin{bmatrix}
\cos(\theta) & -\sin(\theta) & 0 & 0 \\
\sin(\theta) & \cos(\theta) & 0 & 0 \\
0 & 0 & 1 & z \\
0 & 0 & 0 & 1
\end{bmatrix}.
$$

The value and ranges for z and θ are exported by the properties z, theta, zRange and thetaRange, and the coordinate indices are defined by the constants Z_IDX and THETA_IDX. For rendering, the properties shaftLength and shaftRadius control the size of a shaft drawn about the sliding/rotation axis, using the faceColor rendering property. A demo is provided by artsynth.demos.mech.CylindricalJointDemo.

In addition to the standard constructors described in Section 3.3.3,

```java
CylindricalJoint (bodyA, bodyB, originD, zaxis)
```

creates a cylindrical joint with a specified origin and z axis direction for frame D (in world coordinates), and frames C and D coincident.

3.4.4 Slotted Hinge Joint

The SlottedHingeJoint (Figure 3.16) is a 2 DOF joint that constrains motion between frames C and D to translation along the x axis and rotation about the z axis of D. It implements six constraints and two coordinates x and θ (Table 3.4), to which the joint transform $T_{CD}$ is related by

$$
T_{CD} = \begin{bmatrix}
\cos(\theta) & -\sin(\theta) & 0 & 0 & x \\
\sin(\theta) & \cos(\theta) & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & 1 & 0
\end{bmatrix}.
$$

(3.25)
Table 3.3: Constraints (top) and coordinates (bottom) for the cylindrical joint.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along x</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along y</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts rotation about x</td>
</tr>
<tr>
<td>3</td>
<td>bilateral</td>
<td>restricts rotation about y</td>
</tr>
<tr>
<td>4</td>
<td>unilataral</td>
<td>enforces limits on the $z$ coordinate</td>
</tr>
<tr>
<td>5</td>
<td>unilataral</td>
<td>enforces limits on the $\theta$ coordinate</td>
</tr>
</tbody>
</table>

| 0     | $z$       | translation of $C$ along the $z$ axis            |
| 1     | $\theta$  | rotation of $C$ about the $z$ axis               |

Table 3.4: Constraints (top) and coordinates (bottom) for the slotted hinge joint.

The value and ranges for $x$ and $\theta$ are exported by the properties $x$, theta, xRange and thetaRange, and the coordinate indices are defined by the constants $X\_IDX$ and $\Theta\_IDX$. For rendering, the properties shaftLength and shaftRadius control the size of a shaft drawn about the rotation axis, while slotWidth and slotDepth control the width and depth of a slot drawn along the sliding ($x$) axis; both are drawn using the faceColor rendering property. When rendering the slot, its bounds along the $x$ axis are set to $xRange$ by default. However, this may be too large, particularly if $xRange$ is unbounded. As an alternate, the property slotRange will be used instead if its range (i.e., the upper bound minus the lower bound) exceeds 0. A demo of SlottedHingeJoint is provided by artisynth.demos.mech.SlottedHingeJointDemo.

In addition to the standard constructors described in Section 3.3.3,

```java
SlottedHingeJoint (bodyA, bodyB, originD, zaxis)
```

creates a slotted hinge joint with a specified origin and $z$-axis direction for frame D (in world coordinates), and frames C and D coincident.

3.4.5 Universal Joint

The UniversalJoint (Figure 3.17) is a 2 DOF joint that allows C two rotational degrees of freedom with respect to D: a roll rotation $\theta$ about D’s $z$ axis, followed by a pitch rotation $\phi$ about the rotated $y'$ axis. It implements six constraints and
the two coordinates $\theta$ and $\phi$ (Table 3.5), to which the joint transform $T_{CD}$ is related by

$$
T_{CD} = \begin{pmatrix}
    c_{\theta}c_{\phi} & -s_{\theta} & c_{\theta}s_{\phi} & 0 \\
    s_{\theta}c_{\phi} & c_{\theta} & s_{\theta}s_{\phi} & 0 \\
    -s_{\phi} & 0 & c_{\phi} & 0 \\
    0 & 0 & 0 & 1
\end{pmatrix},
$$

where

$$
c_{\theta} \equiv \cos(\theta), s_{\theta} \equiv \sin(\theta), c_{\phi} \equiv \cos(\phi), s_{\phi} \equiv \sin(\phi).
$$

The value and ranges for $\theta$ and $\phi$ are exported by the properties roll, pitch, rollRange and pitchRange, and the coordinate indices are defined by the constants ROLL_IDX and PITCH_IDX. For rendering, the properties shaftLength and shaftRadius control the size of shafts drawn about the roll and pitch axes, while jointRadius specifies the radius of a ball drawn around the origin of $D$; both are drawn using the faceColor rendering property. A demo is provided by artisynth.demos.mech.UniversalJointDemo.

### 3.4.6 Skewed Universal Joint

The SkewedUniversalJoint (Figure 3.18) is a version of the universal joint in which the pitch axis is skewed relative to its nominal direction by an angle $\alpha$. More precisely, let $x'$ and $y'$ be the $x$ and $y$ axes of $C$ after the initial roll rotation. For a regular universal joint, the pitch axis is $y'$, whereas for a skewed universal joint it is $y'$ rotated by $\alpha$ clockwise about $x'$. The joint still has 2 DOF, but the space of allowed rotations is reduced.

The constraints and the coordinates are the same as for the universal joint, although the relationship between $T_{CD}$ is now more complicated. With $c_{\theta}, s_{\theta}, c_{\phi}$, and $s_{\phi}$ defined as for the universal joint, $T_{CD}$ is given by

$$
T_{CD} = \begin{pmatrix}
    c_{\theta}c_{\phi} - s_{\theta}s_{\alpha}s_{\phi} & -s_{\theta} & -s_{\alpha}c_{\theta}s_{\phi} & c_{\alpha}(c_{\theta}s_{\phi} - s_{\alpha}s_{\theta}v_{\phi}) & 0 \\
    s_{\theta}c_{\phi} + c_{\theta}s_{\alpha}s_{\phi} & c_{\theta} & s_{\alpha}c_{\theta}s_{\phi} & c_{\alpha}(s_{\theta}s_{\phi} + s_{\alpha}c_{\theta}v_{\phi}) & 0 \\
    -s_{\phi} & 0 & c_{\phi} & 0 & 0 \\
    0 & 0 & 0 & s_{\alpha}^2 + c_{\alpha}^2 & 0 \\
    0 & 0 & 0 & 0 & 1
\end{pmatrix},
$$

where

$$
c_{\alpha} \equiv \cos(\alpha), \quad s_{\alpha} \equiv \sin(\alpha), \quad v_{\phi} \equiv 1 - c_{\phi}, \quad \beta \equiv c_{\alpha}^2 + s_{\alpha}^2 c_{\phi}.
$$
Figure 3.18: Left: diagram for a skewed universal joint, showing the pitch axis (dotted line) skewed by an angle $\alpha$ relative to its nominal direction along the $y'$ axis. Right: demo model with skew angle of $30^\circ$.

Rendering is controlled using the properties `shaftLength`, `shaftRadius` and `jointRadius` in the same way as for the `UniversalJoint`. A demo is provided by calling `artisynth.demos.mech.UniversalJointDemo` with the model arguments `-skew <angDeg>`, where `<angDeg>` is the desired skew angle in degrees.

Constructors for skewed universal joints take the standard forms described in Section 3.3.3, with an additional argument at the end indicating the skew angle:

```python
SkewedUniversalJoint(bodyA, TCA, bodyB, TCB, skewAngle)
SkewedUniversalJoint(bodyA, bodyB, TDW, skewAngle)
SkewedUniversalJoint(bodyA, bodyB, TCW, TDW, skewAngle)
```

In addition, the constructor

```python
SkewedUniversalJoint(bodyA, bodyB, originD, rollAxis, pitchAxis)
```

creates a skewed universal joint specifying the origin of frame D together with the directions of the roll and pitch axes (in world coordinates). Frames C and D are coincident and the skew angle is inferred from the angle between the axes.

### 3.4.7 Gimbal Joint

Figure 3.19: Coordinate frames (left; rotation angles not shown) and demo model (right) for the gimbal joint.

The GimbalJoint (Figure 3.19) is a 3 DOF spherical joint that anchors the origins of C and D together but otherwise allows C complete rotational freedom. The rotational degrees of freedom are parameterized by three roll-pitch-yaw angles, denoted by $\theta, \phi, \psi$, which define a rotation $\theta$ about D’s $z$ axis, followed by a second rotation $\phi$ about the rotated $y'$ axis, followed by a third rotation $\psi$ about the final $x''$ axis. It implements six constraints and the three coordinates
\( \theta, \phi, \psi \) (Table 3.6), to which the joint transform \( \mathbf{T}_{CD} \) is related by

\[
\mathbf{T}_{CD} = \begin{pmatrix}
    c_\theta c_\phi & c_\theta s_\phi s_\psi - s_\theta c_\psi & c_\phi s_\theta - s_\phi c_\theta s_\psi + s_\theta c_\psi & 0 \\
    s_\theta c_\phi & s_\theta s_\phi s_\psi + c_\theta c_\psi & s_\phi s_\theta c_\psi - c_\theta s_\psi & 0 \\
    -s_\phi & c_\phi s_\psi & c_\phi c_\psi & 0 \\
    0 & 0 & 0 & 1
\end{pmatrix},
\]

where

\[
c_\theta \equiv \cos(\theta), \quad s_\theta \equiv \sin(\theta), \quad c_\phi \equiv \cos(\phi), \quad s_\phi \equiv \sin(\phi), \quad c_\psi \equiv \cos(\psi), \quad s_\psi \equiv \sin(\psi).
\]

The value and ranges for \( \theta, \phi, \psi \) are exported by the properties roll, pitch, yaw, rollRange, pitchRange, and yawRange, and the coordinate indices are defined by the constants ROLL_IDX, PITCH_IDX, and YAW_IDX. For rendering, the property jointRadius specifies the radius of a ball drawn around the origin of D, using the faceColor rendering property. A demo is provided by artisynth.demos.mech.GimbalJointDemo.

In addition to the standard constructors described in Section 3.3.3,

\begin{verbatim}
GimbalJoint (bodyA, bodyB, originD)
\end{verbatim}

creates a gimbal joint with a specified origin for frame D (in world coordinates), and frames C and D coincident and world aligned.

The constraints implementing GimbalJoint are designed so that it is immune to gimbal lock, in which a degree of freedom is lost when \( \phi = \pm \pi/2 \). However, the coordinate values themselves are not immune to this singularity, and neither are the unilateral constraints which enforce limits on their values. Therefore, if coordinate limits are implemented, the joint should be deployed so as try and avoid pitch values near \( \pm \pi/2 \).

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along x</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along y</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts translation along z</td>
</tr>
<tr>
<td>3</td>
<td>unilaterial</td>
<td>enforces limits on the roll coordinate</td>
</tr>
<tr>
<td>4</td>
<td>unilaterial</td>
<td>enforces limits on the pitch coordinate</td>
</tr>
<tr>
<td>5</td>
<td>unilaterial</td>
<td>enforces limits on the yaw coordinate</td>
</tr>
<tr>
<td>0</td>
<td>( \theta ) (roll)</td>
<td>first rotation of C about the z axis of D</td>
</tr>
<tr>
<td>1</td>
<td>( \phi ) (pitch)</td>
<td>second rotation of C about the rotated y' axis</td>
</tr>
<tr>
<td>2</td>
<td>( \psi ) (yaw)</td>
<td>third rotation of C about the final x'' axis</td>
</tr>
</tbody>
</table>

Table 3.6: Constraints (top) and coordinates (bottom) for the gimbal joint.

### 3.4.8 Spherical Joint

![Figure 3.20: Left: coordinate frames of the spherical joint, showing the tilt angle \( \phi \) between the z axes of C and D. Right: demo model for the spherical joint.](image)

The SphericalJoint (Figure 3.20) is a 3 DOF spherical joint that, like GimbalJoint, anchors the origins of C and D together but otherwise allows C complete rotational freedom. SphericalJoint does not implement any coordinates, and so is conceptually more like a ball joint. However, it does provide two choices for limiting its rotation:
• A limit on the tilt angle $\phi$ between the $z$ axes of D and C, such that

$$\phi \leq \phi_{\text{max}}.$$ 

(3.26)

This is intended to emulate the limit imposed by a ball joint socket.

• A limit on the total rotation, defined as follows: Let $(u, \theta)$ be the axis-angle representation of the rotation matrix of $T_{CD}$, normalized such that $\theta \geq 0$ and $\|u\| = 1$, and let $r_{\text{max}}$ be a three-vector giving maximum rotation angles with $x$, $y$, and $z$ components. Then $\theta$ is constrained by

$$\theta \leq \|r_{\text{max}} \circ u\|,$$

(3.27)

where $\circ$ denotes the element-wise product. If the components of $r_{\text{max}}$ are set to a uniform value $\theta_{\text{max}}$, this simplifies to

$$\theta \leq \theta_{\text{max}}.$$

These limits can be enabled by setting the joint’s properties $\text{isTiltLimited}$ and $\text{isRotationLimited}$, respectively, where enabling one disables the other. The limit values $\phi_{\text{max}}$ and $r_{\text{max}}$ are managed using the properties $\text{maxTilt}$ and $\text{maxRotation}$, and setting either automatically enables tilt or rotation limiting, as appropriate. Finally, the tilt angle $\phi$ can be queried using the (read-only) $\text{tilt}$ property. For rendering, the property $\text{jointRadius}$ specifies the radius of a ball drawn around the origin of D, using the $\text{faceColor}$ rendering property. A demo of the SphericalJoint is provided by $\text{artisynth.demos.mech.SphericalJointDemo}$.

In addition to the standard constructors described in Section 3.3.3,

$$\text{SphericalJoint (bodyA, bodyB, originD)}$$

creates a spherical joint with a specified origin for frame D (in world coordinates), and frames C and D coincident and world aligned.

One should use the rotation limit with some caution, as the orientations which it prohibits can be somewhat hard to predict, particularly when $r_{\text{max}}$ has non-uniform values.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along $x$</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along $y$</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts translation along $z$</td>
</tr>
<tr>
<td>3</td>
<td>unilataral</td>
<td>enforces either the “tilt” or “rotation” limits</td>
</tr>
</tbody>
</table>

Table 3.7: Constraints for the spherical joint.

### 3.4.9 Planar Joint

![Planar Joint Diagram](image)

Figure 3.21: Coordinate frames (left) and demo model (right) for the planar joint.

The PlanarJoint (Figure 3.21) is a 3 DOF joint that constrains C to translation in the $x$-$y$ plane and rotation about the $z$ axis of D. It implements six constraints and three coordinates $x$, $y$ and $\theta$ (Table 3.8), to which the joint transform $T_{CD}$ is
related by

\[
T_{CD} = \begin{pmatrix}
\cos(\theta) & -\sin(\theta) & 0 & x \\
\sin(\theta) & \cos(\theta) & 0 & y \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{pmatrix}.
\]

The value and ranges for \(x\), \(y\) and \(\theta\) are exported by the properties \(x\), \(y\), \(\text{theta}\), \(\text{xRange}\), \(\text{yRange}\) and \(\text{thetaRange}\), and the coordinate indices are defined by the constants \(\text{X_IDX}\), \(\text{Y_IDX}\) and \(\text{THETA_IDX}\). A planar joint can be rendered as a square centered on the origin of D, using face rendering properties and with a size given by the \(\text{planeSize}\) property. For example,

```java
PlanarJoint joint;
...
joint.setPlaneSize(5.0);
RenderProps.setFaceColor(joint, Color.LIGHT_GRAY);
```

will cause \(\text{joint}\) to be drawn as a light grey square with size 5.0. The default value of \(\text{planeSize}\) is 0, so drawing the plane is disabled by default. Also, the default \(\text{faceStyle}\) rendering property for \(\text{PlanarConnector}\) is set to \(\text{FRONT_AND_BACK}\), so that the plane (when drawn) can be seen from both sides. A shaft about the rotation axis can also be drawn, as controlled by the properties \(\text{shaftLength}\) and \(\text{shaftRadius}\) and using the \(\text{faceColor}\) rendering property. A demo is provided by \(\text{artisynth.demos.mech.PlanarJointDemo}\).

In addition to the standard constructors described in Section 3.3.3,

```java
PlanarJoint (bodyA, bodyB, originD, zaxis)
```

creates a planar joint with a specified origin and \(z\) axis direction for frame D (in world coordinates), and frames C and D coincident.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along (z)</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts rotation about (x)</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts rotation about (y)</td>
</tr>
<tr>
<td>3</td>
<td>unilataral</td>
<td>enforces limits on the (x) coordinate</td>
</tr>
<tr>
<td>4</td>
<td>unilataral</td>
<td>enforces limits on the (y) coordinate</td>
</tr>
<tr>
<td>5</td>
<td>unilataral</td>
<td>enforces limits on the (\theta) coordinate</td>
</tr>
<tr>
<td>0</td>
<td>(x)</td>
<td>translation of C along the (x) axis of D</td>
</tr>
<tr>
<td>1</td>
<td>(y)</td>
<td>translation of C along the (y) axis of D</td>
</tr>
<tr>
<td>2</td>
<td>(\theta)</td>
<td>rotation of C about the (z) axis of D</td>
</tr>
</tbody>
</table>

Table 3.8: Constraints (top) and coordinates (bottom) for the planar joint.

### 3.4.10 Planar Translation Joint

![Coordinate frames and demo model](image)

Figure 3.22: Coordinate frames (left) and demo model (right) for the planar translation joint.
**Table 3.9: Constraints (top) and coordinates (bottom) for the planar translation joint.**

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along z</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts rotation about x</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts rotation about y</td>
</tr>
<tr>
<td>3</td>
<td>bilateral</td>
<td>restricts rotation about z</td>
</tr>
<tr>
<td>4</td>
<td>unilateral</td>
<td>enforces limits on the x coordinate</td>
</tr>
<tr>
<td>5</td>
<td>unilateral</td>
<td>enforces limits on the y coordinate</td>
</tr>
<tr>
<td>0</td>
<td>x</td>
<td>translation of C along the x axis of D</td>
</tr>
<tr>
<td>1</td>
<td>y</td>
<td>translation of C along the y axis of D</td>
</tr>
</tbody>
</table>

The **PlanarTranslationJoint** (Figure 3.22) is a 2 DOF joint that is the same as the planar joint without rotation: C is restricted to translation in the x-y plane of D. It implements six constraints and two coordinates x and y (Table 3.9), to which the joint transform $T_{CD}$ is related by

$$T_{CD} = \begin{pmatrix} 1 & 0 & 0 & x \\ 0 & 1 & 0 & y \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}.$$

The value and ranges for x and y are exported by the properties x, y, xRange and yRange, and the coordinate indices are defined by the constants X_IDX and Y_IDX. A planar translation joint can be rendered as a square centered on the origin of D, using face rendering properties and with a size given by the planeSize property, in the same way as described for **PlanarJoint**. A demo is provided by artisynth.demos.mech.PlanarJointDemo.

In addition to the standard constructors described in Section 3.3.3, the **PlanarTranslationJoint**

```java
PlanarTranslationJoint (bodyA, bodyB, originD, zaxis)
```

creates a planar translation joint with a specified origin and z axis direction for frame D (in world coordinates), and frames C and D coincident.

### 3.4.11 Solid Joint

The **SolidJoint** is a 0 DOF joint that rigidly constrains C to D. It implements six constraints and no coordinates (Table 3.10) and the resulting $T_{CD}$ is the identity.

There aren’t normally many uses for solid joints. If one wishes to create a complex rigid body by joining together a variety of shapes, this can be done more efficiently by making these shapes mesh components of a single rigid body (Section 3.2.9).

**Table 3.10: Constraints for the solid joint.**

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral</td>
<td>restricts translation along x</td>
</tr>
<tr>
<td>1</td>
<td>bilateral</td>
<td>restricts translation along y</td>
</tr>
<tr>
<td>2</td>
<td>bilateral</td>
<td>restricts rotation about z</td>
</tr>
<tr>
<td>3</td>
<td>bilateral</td>
<td>restricts rotation about x</td>
</tr>
<tr>
<td>4</td>
<td>bilateral</td>
<td>restricts rotation about y</td>
</tr>
<tr>
<td>5</td>
<td>bilateral</td>
<td>restricts rotation about z</td>
</tr>
</tbody>
</table>

### 3.4.12 Planar Connector

The **PlanarConnector** (Figure 3.23) is a 5 DOF connector that attaches the origin of C to the x-y plane of D. C is completely free to rotate, and to translate within the x-y plane. Only motion in the z direction is restricted. **PlanarConnector** implements one constraint and has no coordinates (Table 3.11).
A PlanarConnector constrains a point on body A (located at the origin of C) to move within a plane on body B. Several planar connectors can be employed to constrain body motions in more complicated ways, although one must be careful to avoid overconstraining the system. The connector can also be configured to function unilaterally, via its unilateral property, in which case the point is constrained to lie in the half-space defined by $z \geq 0$ with respect to D. Several unilateral PlanarConnectors can therefore be used to implement a cheap and approximate collision mechanism with fixed collision points.

When set to function unilaterally, overconstraining the system is not an issue because of the way in which ArtiSynth solves unilateral constraints.

A planar connector can be rendered as a square centered on the origin of D, using face rendering properties and with a size given by the planeSize property. The point attached to A can also be rendered using point rendering properties. For example,

```plaintext
PlanarConnector connector;
...
connector.setPlaneSize (5.0);
RenderProps.setFaceColor (connector, Color.LIGHT_GRAY);
RenderProps.setSphericalPoints (connector, 0.1, Color.BLUE);
```

will cause connector to be drawn as a light grey square with size 5, and for the point on body A to be drawn as a blue sphere with radius 0.1. The default value of planeSize is 0, so drawing the plane is disabled by default. Also, the default faceStyle rendering property for PlanarConnector is set to FRONT_AND_BACK, so that the plane (when drawn) can be seen from both sides.

Constructors for the PlanarConnector include

```plaintext
PlanarConnector (bodyA, pCA, bodyB, TDB)
PlanarConnector (bodyA, pCA, TDW)
PlanarConnector (bodyA, bodyA, TDW)
```

where pCA gives the connection point of body A with respect to frame A, TDB gives the transform from frame D to frame B, and TDW gives the transform from frame D to world.

<table>
<thead>
<tr>
<th>Index</th>
<th>type/name</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>bilateral or unilateral</td>
<td>restricts translation along $z$</td>
</tr>
</tbody>
</table>

Table 3.11: Constraints for the planar connector.
3.4.13 Segmented Planar Connector

The \texttt{SegmentedPlanarConnector} (Figure 3.24) is a 5 DOF connector that generalizes \texttt{PlanarConnector} to a piecewise linear surface, to which the origin of \texttt{C} is constrained while \texttt{C} is otherwise completely free to rotate. The surface is specified by a sequence of 2D points defining a piecewise linear curve in the x-z plane of \texttt{D} (Figure 3.24, left). This curve does not need to be a function; the segment nearest to \texttt{C} is the one used to enforce the constraint at any given time. The surface has infinite extent and is extrapolated beyond the first and last segments. It implements one constraint and has no coordinates (Table 3.12).

By appropriate choice of segments, a \texttt{SegmentedPlanarConnector} can approximate any surface defined by a curve in the x-z plane. As with \texttt{PlanarConnector}, it can also be configured as unilateral, constraining the origin of \texttt{C} to lie on the side of the surface defined by the normal vectors \( \mathbf{n}_k \) of each segment \( k \). If \( \mathbf{p}_{k-1} \) and \( \mathbf{p}_k \) are the points in the x-z plane defining the \( k \)-th segment, and \( \hat{y} \) is the y axis unit vector, then \( \mathbf{n}_k \) is given by

\[
\mathbf{n}_k = \frac{\mathbf{u} \times \hat{y}}{\|\mathbf{u} \times \hat{y}\|}, \quad \mathbf{u} \equiv \mathbf{p}_k - \mathbf{p}_{k-1}.
\] (3.28)

The properties controlling the rendering of a segmented planar connector are the same as for a planar connector, with each of the individual plane segments drawn as a rectangle whose length along the y axis is controlled by \texttt{planeSize}.

Constructors for a \texttt{SegmentedPlanarConnector} are analogous to those used for \texttt{PlanarConnector},

\begin{verbatim}
SegmentedPlanarConnector (bodyA, pCA, bodyB, TDB, segs)
SegmentedPlanarConnector (bodyA, pCA, TDW, segs)
SegmentedPlanarConnector (bodyA, bodyA, TDW, segs)
\end{verbatim}

where \texttt{segs} is an additional argument of type \texttt{double[]} giving the 2D coordinates defining the segments in the x-z plane.

3.4.14 Legacy Joints

ArtiSynth maintains three legacy joint for compatibility with earlier software:

- \texttt{RevoluteJoint} is identical to the \texttt{HingeJoint}, except that its coordinate \( \theta \) is oriented \textit{clockwise} about the \textit{z} axis instead of \textit{counter-clockwise}. Rendering is also done differently, with shafts about the rotation axis drawn using line rendering properties.
• **RollPitchJoint** is identical to the **UniversalJoint**, except that its roll-pitch coordinates $\theta, \phi$ are computed with respect to the rotation $R_{DC}$ from frame D to C, instead of the rotation $R_{CD}$ from frame C to D. Rendering is also done differently, with shafts along the roll and pitch axes drawn using line rendering properties, and the ball around the origin of D drawn using point rendering properties.

• **SphericalRpyJoint** is identical to the **GimbalJoint**, except that its roll-pitch-yaw coordinates $\theta, \phi, \psi$ are computed with respect to the rotation $R_{DC}$ from frame D to C, instead of the rotation $R_{CD}$ from frame C to D. Rendering is also done differently, with the ball around the origin of D drawn using point rendering properties.

### 3.5 Frame springs

Another way to connect two rigid bodies together is to use a *frame spring*, which is a six dimensional spring that generates restoring forces and moments between coordinate frames.

#### 3.5.1 Frame spring coordinate frames

![Frame Spring Diagram](image)

Figure 3.25: A frame spring connecting two coordinate frames D and C.

The basic idea of a frame spring is shown in Figure 3.25. It generates restoring forces and moments on two frames C and D which are a function of $T_{DC}$ and $\dot{v}_{DC}$ (the spatial velocity of frame D with respect to frame C).

Decomposing forces into stiffness and damping terms, the force $f_C$ and moment $\tau_C$ acting on C can be expressed as

$$
\begin{align*}
  f_C &= f_k(T_{DC}) + f_d(\dot{v}_{DC}) \\
  \tau_C &= \tau_k(T_{DC}) + \tau_d(\dot{v}_{DC}).
\end{align*}
$$

(3.29)

where the translational and rotational forces $f_k, f_d, \tau_k, \tau_d$ are general functions of $T_{DC}$ and $\dot{v}_{DC}$.

The forces acting on D are equal and opposite, so that

$$
\begin{align*}
  f_D &= -f_C, \\
  \tau_D &= -\tau_C.
\end{align*}
$$

(3.30)

If frames C and D are attached to a pair of rigid bodies A and B, then a frame spring can be used to connect them in a manner analogous to a joint. As with joints, C and D generally do not coincide with the body frames, and are instead offset from them by fixed transforms $T_{CA}$ and $T_{DB}$ (Figure 3.26).

#### 3.5.2 Frame materials

The restoring forces (3.29) generated in a frame spring depend on the *frame material* associated with the spring. Frame materials are defined in the package `artisynth.core.materials`, and are subclassed from `FrameMaterial`. The most basic type of material is a **LinearFrameMaterial**, in which the restoring forces are determined from

$$
\begin{align*}
  f_C &= K_t s_{DC} + D_t v_{DC} \\
  \tau_C &= K_r \dot{\theta}_{DC} + D_r \omega_{DC}
\end{align*}
$$
Figure 3.26: A frame spring connecting two rigid bodies A and B.

where \( \hat{\theta}_{DC} \) gives the small angle approximation of the rotational components of \( X_{DC} \) with respect to the \( x, y, \) and \( z \) axes, and

\[
K_r = \begin{pmatrix} k_{rx} & 0 & 0 \\ 0 & k_{ry} & 0 \\ 0 & 0 & k_{rz} \end{pmatrix}, \quad D_r = \begin{pmatrix} d_{rx} & 0 & 0 \\ 0 & d_{ry} & 0 \\ 0 & 0 & d_{rz} \end{pmatrix},
\]

are the stiffness and damping matrices. The diagonal values defining each matrix are stored in the 3-dimensional vectors \( k_t, k_r, d_t, \) and \( d_r \) which are exposed as the stiffness, rotaryStiffness, damping, and rotaryDamping properties of the material. Each of these specifies stiffness or damping values along or about a particular axis. Specifying different values for different axes will result in anisotropic behavior.

Other frame materials offering nonlinear behavior may be defined in artisynth.core.materials.

### 3.5.3 Creating frame springs

Frame springs are implemented by the class FrameSpring. Creating a frame spring generally involves instantiating this class, and then setting the material, the bodies A and B, and the transforms \( T_{CA} \) and \( T_{DB} \).

A typical construction sequence might look like this:

```java
FrameSpring spring = new FrameSpring("springA");
spring.setMaterial(new LinearFrameMaterial (kt, kr, dt, dr));
spring.setFrames(bodyA, bodyB, TDW);
```

The material is set using `setMaterial()`. The example above uses a `LinearFrameMaterial`, created with a constructor that sets \( k_t, k_r, d_t, \) and \( d_r \) to uniform Isotropic values specified by \( kt, kr, dt, \) and \( dr \).

The bodies and transforms can be set in the same manner as for joints (Section 3.3.3), with the methods `setFrames(bodyA, bodyB, TDW)` and `setFrames(bodyA, TCA, bodyB, TDB)` assuming the role of the `setBodies()` methods used for joints. The former takes D specified in world coordinates and computes \( T_{CA} \) and \( T_{DB} \) assuming that there is no initial spring displacement (i.e., that \( T_{DC} = I \)), while the latter allows \( T_{CA} \) and \( T_{DB} \) to be specified explicitly with \( T_{DC} \) assuming whatever value is implied.

Frame springs and joints are often placed together, using the same transforms \( T_{CA} \) and \( T_{DB} \), with the spring providing restoring forces to help keep the joint within prescribed bounds.

As with joints, a frame spring can be connected to only a single body, by specifying `frameB` as `null`. Frame B is then taken to be the world coordinate frame W.
Figure 3.27: LumbarFrameSpring model loaded into ArtiSynth.

3.5.4 Example: two bodies connected by a frame spring

A simple model showing two simplified lumbar vertebrae, modeled as rigid bodies and connected by a frame spring, is defined in

artisynth.demos.tutorial.LumbarFrameSpring

The definition for the entire model class is shown here:

```java
package artisynth.demos.tutorial;

import java.io.IOException;
import java.io.File;
import java.awt.Color;
import artisynth.core.modelbase.*;
import artisynth.core.mechmodels.*;
import artisynth.core.materials.*;
import artisynth.core.workspace.RootModel;
import maspack.matrix.*;
import maspack.geometry.*;
import maspack.render.*;
import maspack.util.PathFinder;

/**
 * Demo of two rigid bodies connected by a 6 DOF frame spring
 */
public class LumbarFrameSpring extends RootModel {
    double density = 1500;
    // path from which meshes will be read
    private String geometryDir = PathFinder.getSourceRelativePath (LumbarFrameSpring.class, "../mech/geometry/");

    // create and add a rigid body from a mesh
    public RigidBody addBone (MechModel mech, String name) throws IOException {
        PolygonalMesh mesh = new PolygonalMesh (new File (geometryDir+name+".obj"));
        RigidBody rb = RigidBody.createFromMesh (name, mesh, density, "scale=*/1");
        mech.addRigidBody (rb);
        return rb;
    }
}
```
public void build (String[] args) throws IOException {
    // create mech model and set it’s properties
    MechModel mech = new MechModel("mech");
    mech.setGravity (0, 0, -1.0);
    mech.setFrameDamping (0.10);
    mech.setRotaryDamping (0.001);
    addModel (mech);

    // create two rigid bodies and second one to be fixed
    RigidBody lumbar1 = addBone (mech, "lumbar1");
    RigidBody lumbar2 = addBone (mech, "lumbar2");
    lumbar1.setPose (new RigidTransform3d (-0.016, 0.039, 0));
    lumbar2.setDynamic (false);

    // flip entire mech model around
    mech.transformGeometry (new RigidTransform3d (0, 0, 0, 0, 0, Math.toRadians (90)));

    // create and add the frame spring
    FrameSpring spring = new FrameSpring (null);
    spring.setMaterial (new LinearFrameMaterial (/* ktrans=*/100, /* krot=*/0.01, /* dtrans=*/0, /* drot=*/0));
    spring.setFrames (lumbar1, lumbar2, lumbar1.getPose());
    mech.addFrameSpring (spring);

    // set render properties for components
    RenderProps.setLineColor (spring, Color.RED);
    RenderProps.setLineWidth (spring, 3);
    spring.setAxisLength (0.02);
    RenderProps.setFaceColor (mech, new Color (238, 232, 170)); // bone color
}

For convenience, the code to create and add each vertebrae is wrapped into the method addBone() defined at lines 27-32. This method takes two arguments: the MechModel to which the bone should be added, and the name of the bone. Surface meshes for the bones are located in .obj files located in the directory ..//mech/geometry relative to the source directory for the model itself. PathFinder.getSourceRelativePath() is used to find a proper path to this directory (see Section 2.6) given the model class type (LumbarFrameSpring.class), and this is stored in the static string geometryDir. Within addBone(), the directory path and the bone name are used to create a path to the bone mesh itself, which is in turn used to create a PolygonalMesh (line 28). The mesh is then used in conjunction with a density to create a rigid body which is added to the MechModel (lines 29-30) and returned.

The build() method begins by creating and adding a MechModel, specifying a low value for gravity, and setting the rigid body damping properties frameDamping and rotaryDamping (lines 37-41). (The damping parameters are needed here because the frame spring itself is created with no damping.) Rigid bodies representing the vertebrae lumbar1 and lumbar2 are then created by calling addBone() (lines 44-45), lumbar1 is translated by setting the origin of its pose to (−0.016, 0.039, 0)T, and lumbar2 is set to be fixed by making it non-dynamic (line 47).

At this point in the construction, if the model were to be loaded, it would appear as in Figure 3.28. To change the viewpoint to that seen in Figure 3.27, we rotate the entire model about the x axis (line 50). This is done using transformGeometry(X), which transforms the geometry of an entire model using a rigid or affine transform. This method is described in more detail in Section 4.6.

The frame spring is created and added at lines 54-59, using the methods described in Section 3.5.3, with frame D set to the (initial) pose of lumbar1.

Render properties are set starting at line 62. By default, a frame spring renders as a pair of red, green, blue coordinate axes showing frames C and D, along with a line connecting them. The line width and the color of the connecting line are controlled by the line render properties lineWidth and lineColor, while the length of the coordinate axes is controlled by the special frame spring property axisLength.

To run this example in ArtiSynth, select All demos > tutorial > LumbarFrameSpring from the Models menu. The model should load and initially appear as in Figure 3.27. Running the model (Section 1.5.3) will cause lumbar1 to fall slightly
under gravity until the frame spring arrests the motion. To get a sense of the spring’s behavior, one can interactively apply forces to lumbar1 using the pull tool (see the section “Pull Manipulation” in the ArtiSynth User Interface Guide).

### 3.6 Attachments

ArtiSynth provides the ability to rigidly attach dynamic components to other dynamic components, allowing different parts of a model to be connected together. Attachments are made by adding to a MechModel special attachment components that manage the attachment physics as described briefly in Section 1.2.

#### 3.6.1 Point attachments

Point attachments allow particles and other point-based components to be attached to other, more complex components, such as frames, rigid bodies, or finite element models (Section 6.4). Point attachments are implemented by creating attachment components that are instances of PointAttachment. Modeling applications do not generally handle the attachment components directly, but instead create them implicitly using the following MechModel method:

```
attachPoint (Point p1, PointAttachable comp);
```

This attaches a point p1 to any component which implements the interface PointAttachable, indicating that it is capable creating an attachment to a point. Components that implement PointAttachable currently include rigid bodies, particles, and finite element models. The attachment is created based on the the current position of the point and component in question. For attaching a point to a rigid body, another method may be used:

```
attachPoint (Point p1, RigidBody body, Point3d loc);
```

This attaches p1 to body at the point loc specified in body coordinates. Finite element attachments are discussed in Section 6.4.

Once a point is attached, it will be in the attached state, as described in Section 3.1.3. Attachments can be removed by calling

```
detachPoint (Point p1);
```

#### 3.6.2 Example: model with particle attachments

A model illustrating particle-particle and particle-rigid body attachments is defined in
and most of the code is shown here:

```java
public Particle addParticle (MechModel mech, double x, double y, double z) {
    // create a particle at x, y, z and add it to mech
    Particle p = new Particle(/* name=*/null, /* mass=*/.1, x, y, z);
    mech.addParticle(p);
    return p;
}

public AxialSpring addSpring (MechModel mech, Particle p1, Particle p2) {
    // create a spring connecting p1 and p2 and add it to mech
    AxialSpring spr = new AxialSpring(/* name=*/null, /* restLength=*/0);
    spr.setMaterial(new LinearAxialMaterial(/* k=*/20, /* d=*/10));
    spr.setPoints(p1, p2);
    mech.addAxialSpring(spr);
    return spr;
}

public void build (String[] args) {
    // create MechModel and add to RootModel
    MechModel mech = new MechModel("mech");
    addModel(mech);

    // create the components
    Particle p1 = addParticle(mech, 0, 0, 0.55);
    Particle p2 = addParticle(mech, 0.1, 0, 0.35);
    Particle p3 = addParticle(mech, 0.1, 0, 0.35);
    Particle p4 = addParticle(mech, 0, 0, 0.15);
    addSpring(mech, p1, p2);
    addSpring(mech, p3, p4);

    // create box and set its pose (position/orientation):
    RigidBody box =
        RigidBody.createBox("box", /*wx,wy,wz=*/0.5, 0.3, 0.3, /* density=*/20);
    box.setPose(new RigidTransform3d(/* x,y,z=*/0.2, 0, 0));
    mech.addRigidBody(box);
    p1.setDynamic (false); // first particle set to be fixed

    // set up the attachments
```

Figure 3.29: ParticleAttachment model loaded into ArtiSynth.
The code is very similar to ParticleSpring and RigidBodySpring described in Sections 3.1.2 and 3.2.2, except that two convenience methods, addParticle() and addSpring(), are defined at lines 1-15 to create particles and spring and add them to a MechModel. These are used in the build() method to create four particles and two springs (lines 24-29), along with a rigid body box (lines 31-34). As with the other examples, particle p1 is set to be non-dynamic (line 36) in order to fix it in place and provide a ground.

The attachments are added at lines 39-40, with p2 attached to p3 and p4 connected to the box at the location (0, 0, 0.15) in box coordinates.

Finally, render properties are set starting at line 43. In this example, point and line render properties are set for the entire MechModel instead of individual components. Since render properties are inherited, this will implicitly set the specified render properties in all subcomponents for which these properties are not explicitly set (either locally or in an intermediate ancestor).

To run this example in ArtiSynth, select All demos > tutorial > ParticleAttachment from the Models menu. The model should load and initially appear as in Figure 3.29. Running the model (Section 1.5.3) will cause the box to fall and swing under gravity.

### 3.6.3 Frame attachments

Frame attachments allow rigid bodies and other frame-based components to be attached to other components, including frames, rigid bodies, or finite element models (Section 6.6). Frame attachments are implemented by creating attachment components that are instances of FrameAttachment.

As with point attachments, modeling applications do not generally handle frame attachment components directly, but instead create and add them implicitly using the following MechModel methods:

```java
attachFrame (Frame frame, FrameAttachable comp);
attachFrame (Frame frame, FrameAttachable comp, RigidTransform3d TFW);
```

These attach frame to any component which implements the interface FrameAttachable, indicating that it is capable of creating an attachment to a frame. Components that implement FrameAttachable currently include frames, rigid bodies, and finite element models. For the first method, the attachment is created based on the current position of the frame and component in question. For the second method, the attachment is created so that the initial pose of the frame (in world coordinates) is described by TFW.

Once a frame is attached, it will be in the attached state, as described in Section 3.1.3. Frame attachments can be removed by calling

```java
detachFrame (Frame frame);
```

While it is possible to create composite rigid bodies using FrameAttachments, this is much less computationally efficient (and less accurate) than creating a single rigid body through mesh merging or similar techniques.

### 3.6.4 Example: model with frame attachments

A model illustrating rigidBody-rigidBody and frame-rigidBody attachments is defined in

```java
artisynth.demos.tutorial.FrameBodyAttachment
```
Most of the code is identical to that for RigidBodyJoint as described in Section 3.3.6, except that the joint is further to the left and connects bodyB to ground, rather than to bodyA, and the initial pose of bodyA is changed so that it is aligned vertically. bodyA is then connected to bodyB, and an auxiliary frame is created and attached to bodyA, using code at the end of the build() method as shown here:

```java
public void build (String[] args) {

  // create model mostly similar to RigidBodyJoint ...

  // now connect bodyA to bodyB using a FrameAttachment
  mech.attachFrame (bodyA, bodyB);

  // create an auxiliary frame and add it to the mech model
  Frame frame = new Frame();
  mech.addFrame (frame);

  // set the frames axis length > 0 so we can see it
  frame.setAxisLength (4.0);

  // set the attached frame’s pose to that of bodyA ...
  RigidTransform3d TFW = new RigidTransform3d (bodyA.getPose());

  // ... plus a translation of lenx2/2 along the x axis:
  TFW.mulXyz (lenx2/2, 0, 0);

  // finally, attach the frame to bodyA
  mech.attachFrame (frame, bodyA, TFW);
}
```

To run this example in ArtiSynth, select All demos > tutorial > FrameBodyAttachment from the Models menu. The model should load and initially appear as in Figure 3.29. The frame attached to bodyA is visible in the lower right corner. Running the model (Section 1.5.3) will cause both bodies to fall and swing about the joint under gravity.
Chapter 4

Mechanical Models II

This section provides additional material on building basic multibody-type mechanical models.

4.1 Simulation control properties

Both RootModel and MechModel contain properties that control the simulation behavior.

4.1.1 maxStepSize

One of the most important of these is maxStepSize. By default, simulation proceeds using the maxStepSize value defined for the root model. A MechModel (or any other type of Model) contained in the root model’s models list may also request a smaller step size by specifying a smaller value for its own maxStepSize property. For all models, the maxStepSize may be set and queried using:

```java
void setMaxStepSize (double maxh);
double getMaxStepSize();
```

4.1.2 integrator

Another important simulation property is integrator in MechModel, which determines the type of integrator used for the physics simulation. The value type of this property is the enumerated type MechSystemSolver.Integrator, for which the following values are currently defined:

- **ForwardEuler**
  - First order forward Euler integrator. Unstable for stiff systems.

- **SymplecticEuler**
  - First order symplectic Euler integrator, more energy conserving than forward Euler. Unstable for stiff systems.

- **RungeKutta4**
  - Fourth order Runge-Kutta integrator, quite accurate but also unstable for stiff systems.

- **ConstrainedBackwardEuler**
  - First order backward order integrator. Generally stable for stiff systems.

- **Trapezoidal**
  - Second order trapezoidal integrator. Generally stable for stiff systems, but slightly less so than ConstrainedBackwardEuler.
The term “Unstable for stiff systems” means that the integrator is likely to go unstable in the presence of “stiff” systems, which typically include systems containing finite element models, unless the simulation step size is set to an extremely small value. The default value for integrator is ConstrainedBackwardEuler.

Stiff systems tend to arise in models containing interconnected deformable elements, for which the step size should not exceed the propagation time across the smallest element, an effect known as the Courant-Friedrichs-Lewy (CFL) condition. Larger stiffness and damping values decrease the propagation time and hence the allowable step size.

### 4.1.3 stabilization

Another MechModel simulation property is stabilization, which controls the stabilization method used to correct drift from position constraints and correct interpenetrations due to collisions (Chapter 7). The value type of this property value is the enumerated type MechSystemSolver.PosStabilization, which presently has two values:

**GlobalMass**

Uses only a diagonal mass matrix for the MLCP that is solved to determine the position corrections. This is the default method.

**GlobalStiffness**

Uses a stiffness-corrected mass matrix for the MLCP that is solved to determine the position corrections. Slower than GlobalMass, but more likely to produce stable results, particularly for problems involving FEM collisions.

### 4.2 Units

ArtiSynth is primarily “unitless”, in the sense that it does not define default units for the fundamental physical quantities of time, length, and mass. Although time is generally understood to be in seconds, and often declared as such in method arguments and return values, there is no hard requirement that it be interpreted as seconds. There are no assumptions at all regarding length and mass. Some components may have default parameter values that reflect a particular choice of units, such as MechModel’s default gravity value of \((0, 0, -9.8)\), which is associated with the MKS system, but these values can always be overridden by the application.

Nevertheless, it is important, and up to the application developer to ensure, that units be consistent. For example, if one decides to switch length units from meters to centimeters (a common choice), then all units involving length will have to be scaled appropriately. For example, density, whose fundamental units are \(m/d^3\), where \(m\) is mass and \(d\) is distance, needs to be scaled by \(1/100^3\), or 0.000001, when converting from meters to centimeters.

Table 4.1 lists a number of common physical quantities used in ArtiSynth, along with their associated fundamental units.

### 4.2.1 Scaling units

For convenience, many ArtiSynth components, including MechModel, implement the interface ScalableUnits, which provides the following methods for scaling mass and distance units:

```java
scaleDistance (s); // scale distance units by s
scaleMass (s);     // scale mass units by s
```

A call to one of these methods should cause all physical quantities within the component (and its descendants) to be scaled as required by the fundamental unit relationships as shown in Table 4.1.

Converting a MechModel from meters to centimeters can therefore be easily done by calling

```java
mech.scaleDistance (100);
```

As an example, adding the following code to the end of the build() method in RigidBodySpring (Section 3.2.2)
<table>
<thead>
<tr>
<th>unit</th>
<th>fundamental units</th>
</tr>
</thead>
<tbody>
<tr>
<td>time</td>
<td>( t )</td>
</tr>
<tr>
<td>distance</td>
<td>( d )</td>
</tr>
<tr>
<td>mass</td>
<td>( m )</td>
</tr>
<tr>
<td>velocity</td>
<td>( \frac{d}{t} )</td>
</tr>
<tr>
<td>acceleration</td>
<td>( \frac{d}{t^2} )</td>
</tr>
<tr>
<td>force</td>
<td>( md/t^2 )</td>
</tr>
<tr>
<td>work/energy</td>
<td>( md^2/t^2 )</td>
</tr>
<tr>
<td>torque</td>
<td>( md^2/t^2 ) same as energy (somewhat counter intuitive)</td>
</tr>
<tr>
<td>angular velocity</td>
<td>( 1/t )</td>
</tr>
<tr>
<td>angular acceleration</td>
<td>( 1/t^2 )</td>
</tr>
<tr>
<td>rotational inertia</td>
<td>( md^2 )</td>
</tr>
<tr>
<td>pressure</td>
<td>( m/(dt^2) )</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>( m/(dt^2) )</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>( 1 ) no units; it is a ratio</td>
</tr>
<tr>
<td>density</td>
<td>( m/d^3 )</td>
</tr>
<tr>
<td>linear stiffness</td>
<td>( m/t^2 )</td>
</tr>
<tr>
<td>linear damping</td>
<td>( m/t )</td>
</tr>
<tr>
<td>rotary stiffness</td>
<td>( md^2/t^2 ) same as torque</td>
</tr>
<tr>
<td>rotary damping</td>
<td>( md^2/t )</td>
</tr>
<tr>
<td>mass damping</td>
<td>( 1/t ) used in FemModel</td>
</tr>
<tr>
<td>stiffness damping</td>
<td>( t ) used in FemModel</td>
</tr>
</tbody>
</table>

Table 4.1: Physical quantities and their representation in terms of the fundamental units of mass (\( m \)), distance (\( d \)), and time (\( t \)).

```java
System.out.println("length=");
System.out.println("density=");
System.out.println("gravity=");
mech.scaleDistance(100);
System.out.println("gravity");

System.out.println("scaled length=");
System.out.println("scaled density=");
System.out.println("scaled gravity");
```

will scale the distance units by 100 and print the values of various quantities before and after scaling. The resulting output is:

```
length=0.5
density=20.0
gravity=0.0 0.0 -9.8

scaled length=50.0
scaled density=2.0E-5
scaled gravity=0.0 0.0 -980.0
```

It is important not to confuse scaling units with scaling the actual geometry or mass. Scaling units should change all physical quantities so that the simulated behavior of the model remains unchanged. If the distance-scaled version of `RigidBodySpring` shown above is run, it should behave exactly the same as the non-scaled version.

### 4.3 Render properties

All ArtiSynth components that are renderable maintain a property `renderProps`, which stores a `RenderProps` object that contains a number of subproperties used to control an object’s rendered appearance.

In code, the `renderProps` property for an object can be set or queried using the methods

```java
System.out.println("length=");
System.out.println("density=");
System.out.println("gravity=");
mech.scaleDistance(100);
System.out.println("gravity");

System.out.println("scaled length=");
System.out.println("scaled density=");
System.out.println("scaled gravity");
```
<table>
<thead>
<tr>
<th>property</th>
<th>purpose</th>
<th>usual default value</th>
</tr>
</thead>
<tbody>
<tr>
<td>visible</td>
<td>whether or not the component is visible</td>
<td>true</td>
</tr>
<tr>
<td>alpha</td>
<td>transparency for diffuse colors (range 0 to 1)</td>
<td>1 (opaque)</td>
</tr>
<tr>
<td>shading</td>
<td>shading style: (FLAT, SMOOTH, METAL, NONE)</td>
<td>FLAT</td>
</tr>
<tr>
<td>shininess</td>
<td>shininess parameter (range 0 to 128)</td>
<td>32</td>
</tr>
<tr>
<td>specular</td>
<td>specular color components</td>
<td>null</td>
</tr>
<tr>
<td>faceStyle</td>
<td>which polygonal faces are drawn</td>
<td>FRONT</td>
</tr>
<tr>
<td>faceColor</td>
<td>diffuse color for drawing faces</td>
<td>GRAY</td>
</tr>
<tr>
<td>backColor</td>
<td>diffuse color used for the backs of faces. If null, faceColor is used.</td>
<td>null</td>
</tr>
<tr>
<td>drawEdges</td>
<td>hint that polygon edges should be drawn explicitly</td>
<td>false</td>
</tr>
<tr>
<td>colorMap</td>
<td>color mapping properties (see Section 4.3.3)</td>
<td>null</td>
</tr>
<tr>
<td>normalMap</td>
<td>normal mapping properties (see Section 4.3.3)</td>
<td>null</td>
</tr>
<tr>
<td>bumpMap</td>
<td>bump mapping properties (see Section 4.3.3)</td>
<td>null</td>
</tr>
<tr>
<td>edgeColor</td>
<td>diffuse color for edges</td>
<td>null</td>
</tr>
<tr>
<td>edgeWidth</td>
<td>edge width in pixels</td>
<td>1</td>
</tr>
<tr>
<td>lineStyle</td>
<td>how lines are drawn</td>
<td>LINE</td>
</tr>
<tr>
<td>lineColor</td>
<td>diffuse color for lines</td>
<td>GRAY</td>
</tr>
<tr>
<td>lineWidth</td>
<td>width in pixels when LINE style is selected</td>
<td>1</td>
</tr>
<tr>
<td>lineRadius</td>
<td>radius when CYLINDER or SPINDLE style is selected</td>
<td>1</td>
</tr>
<tr>
<td>pointStyle</td>
<td>how points are drawn</td>
<td>POINT</td>
</tr>
<tr>
<td>pointColor</td>
<td>diffuse color for points</td>
<td>GRAY</td>
</tr>
<tr>
<td>pointSize</td>
<td>point size in pixels when POINT style is selected</td>
<td>1</td>
</tr>
<tr>
<td>pointRadius</td>
<td>sphere radius when SPHERE style is selected</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 4.2: Render properties and their default values.

```java
setRenderProps(RenderProps props); // set render properties
RenderProps getRenderProps(); // get render properties (read-only)
```

Render properties can also be set in the GUI by selecting one or more components and the choosing Set render props ... in the right-click context menu. More details on setting render properties through the GUI can be found in the section “Render properties” in the ArtiSynth User Interface Guide.

For many components, the default value of renderProps is null; i.e., no RenderProps object is assigned by default and render properties are instead inherited from ancestor components further up the hierarchy. The reason for this is because RenderProps objects are fairly large (many kilobytes), and so assigning a unique one to every component could consume too much memory. Even when a RenderProps object is assigned, most of its properties are inherited by default, and so only those properties which are explicitly set will differ from those specified in ancestor components.

### 4.3.1 Render property taxonomy

In general, the properties in RenderProps are used to control the color, size, and style of the three primary rendering primitives: faces, lines, and points. Table 4.2 contains a complete list. Values for the shading, faceStyle, lineStyle and pointStyle properties are defined using the following enumerated types: `Renderer.Shading`, `Renderer.FaceStyle`, `Renderer.PointStyle`, and `Renderer.LineStyle`. Colors are specified using `java.awt.Color`.

To increase and improve their visibility, both the line and point primitives are associated with styles (CYLINDER, SPINDLE, and SPHERE) that allow them to be rendered using 3D surface geometry.

Exactly how a component interprets its render properties is up to the component (and more specifically, up to the rendering method for that component). Not all render properties are relevant to all components, particularly if the rendering does not use all of the rendering primitives. For example, Particle components use only the point primitives and AxialSpring components use only the line primitives. For this reason, some components use subclasses of RenderProps, such as PointRenderProps and LineRenderProps, that expose only a subset of the available render properties. All renderable components provide the method `createRenderProps()` that will create and return a RenderProps object suitable for that component.
4.3.2 Setting render properties

When setting render properties, it is important to note that the value returned by `getRenderProps()` should be treated as read-only and should not be used to set property values. For example, applications should not do the following:

```java
particle.getRenderProps().setPointColor(Color.BLUE);
```

This can cause problems for two reasons. First, `getRenderProps()` will return `null` if the object does not currently have a `RenderProps` object. Second, because `RenderProps` objects are large, ArtiSynth may try to share them between components, and so by setting them for one component, the application may inadvertently set them for other components as well.

Instead, `RenderProps` provides a static method for each property that can be used to set that property’s value for a specific component. For example, the correct way to set `pointColor` is

```java
RenderProps.setPointColor(particle, Color.BLUE);
```

One can also set render properties by calling `setRenderProps()` with a predefined `RenderProps` object as an argument. This is useful for setting a large number of properties at once:

```java
RenderProps props = new RenderProps();
props.setPointColor(Color.BLUE);
props.setPointRadius(2);
props.setPointStyle(RenderProps.PointStyle.SPHERE);
...
particle.setRenderProps(props);
```

For setting each of the color properties within `RenderProps`, one can use either `Color` objects or `float[]` arrays of length 3 giving the RGB values. Specifically, there are methods of the form

```java
props.setXXXColor(Color color)
props.setXXXColor(float[] rgb)
```

as well as the static methods

```java
RenderProps.setXXXColor(Renderable r, Color color)
RenderProps.setXXXColor(Renderable r, float[] rgb)
```

where `XXX` corresponds to `Point`, `Line`, `Face`, `Edge`, and `Back`. For `Edge` and `Back`, both `color` and `rgb` can be given as `null` to clear the indicated color. For the specular color, the associated methods are

```java
props.setSpecular(Color color)
props.setSpecular(float[] rgb)
RenderProps.setSpecular(Renderable r, Color color)
RenderProps.setSpecular(Renderable r, float[] rgb)
```

Note that even though components may use a subclass of `RenderProps` internally, one can always use the base `RenderProps` class to set values; properties which are not relevant to the component will simply be ignored.

Finally, as mentioned above, render properties are inherited. Values set high in the component hierarchy will be inherited by descendant components, unless those descendants (or intermediate components) explicitly set overriding values. For example, a `MechModel` maintains its own `RenderProps` (and which is never null). Setting its `pointColor` property to `RED` will cause all point-related components within that `MechModel` to be rendered as red except for components that set their `pointColor` to a different property.

There are typically three levels in a `MechModel` component hierarchy at which render properties can be set:

- The `MechModel` itself;
- Lists containing components;
• Individual components.

For example, consider the following code:

```java
MechModel mech = new MechModel("mech");

Particle p1 = new Particle(/* name = */null, 2, 0, 0, 0);
Particle p2 = new Particle(/* name = */null, 2, 1, 0, 0);
Particle p3 = new Particle(/* name = */null, 2, 1, 1, 0);

mech.addParticle(p1);
mech.addParticle(p2);
mech.addParticle(p3);

RenderProps.setPointColor(mech, Color.BLUE);
RenderProps.setPointColor(mech.particles(), Color.GREEN);
RenderProps.setPointColor(p3, Color.RED);
```

Setting the MechModel render property `pointColor` to `BLUE` will cause all point-related items to be rendered blue by default. Setting the `pointColor` render property for the particle list (returned by `mech.particles()`) will override this and cause all particles in the list to be rendered green by default. Lastly, setting `pointColor` for `p3` will cause it to be rendered as red.

### 4.3.3 Texture mapping

Render properties can also be set to apply texture mapping to objects containing polygonal meshes in which texture coordinates have been set. Supported is provided for color, normal and bump mapping, although normal and bump mapping are only available under the OpenGL 3 version of the ArtiSynth renderer.

Texture mapping is controlled through the `colorMap`, `normalMap`, and `bumpMap` properties of `RenderProps`. These are composite properties with a default value of `null`, but applications can set them to instances of `ColorMapProps`, `NormalMapProps`, and `BumpMapProps`, respectively, to provide the source images and parameters for the associated mapping. The two most important properties exported by all of these `MapProps` objects are:

- **enabled**
  A boolean indicating whether or not the mapping is enabled.

- **fileName**
  A string giving the file name of the supporting source image.

`NormalMapProps` and `BumpMapProps` also export `scaling`, which scales the x-y components of the normal map or the depth of the bump map. Other exported properties control mixing with underlying colors, and how texture coordinates are both filtered and managed when they fall outside the canonical range \([0, 1]\). Full details on texture mapping and its support by the ArtiSynth renderer are given in the “Rendering” section of the Maspack Reference Manual.

To set up a texture map, one creates an instance of the appropriate `MapProps` object and uses this to set either the `colorMap`, `normalMap`, or `bumpMap` property of `RenderProps`. For a specific renderable, the map properties can be set using the static methods:

```java
void RenderProps.setColorMap (Renderable r, ColorMapProps tprops);
void RenderProps.setNormalMap (Renderable r, NormalMapProps tprops);
void RenderProps.setBumpMap (Renderable r, BumpMapProps tprops);
```

When initializing the `PropMaps` object, it is often sufficient to just set `enabled` to `true` and `fileName` to the full path name of the source image. Normal and bump maps also often require adjustment of their `scaling` properties. The following static methods are available for setting the `enabled` and `fileName` subproperties within a renderable:

```java
void RenderProps.setColorMapEnabled (Renderable r, boolean enabled);
void RenderProps.setColorMapFileName (Renderable r, String fileName);
void RenderProps.setNormalMapEnabled (Renderable r, boolean enabled);
```
void RenderProps.setNormalMapFileName(Renderable r, String fileName);
void RenderProps.setBumpMapEnabled(Renderable r, boolean enabled);
void RenderProps.setBumpMapFileName(Renderable r, String fileName);

Normal and bump mapping only work under the OpenGL 3 version of the ArtiSynth viewer, and also do not work if the shading property of RenderProps is set to NONE or FLAT.

Texture mapping properties can be set within ancestor nodes of the component hierarchy, to allow file names and other parameters to be propagated throughout the hierarchy. However, when this is done, it is still necessary to ensure that the corresponding mapping properties for the relevant descendants are non-null. That’s because mapping properties themselves are not inherited; only their subproperties are. If a mapping property for any given object is null, the associated mapping will be disabled. A non-null mapping property for an object will be created automatically by calling one of the setXXXEnabled() methods listed above. So when setting up ancestor-controlled mapping, one may use a construction like this:

```
RenderProps.setColorMap (ancestor, tprops);
RenderProps.setColorMapEnabled (descendant0, true);
RenderProps.setColorMapEnabled (descendant1, true);
```

Then colorMap subproperties set within ancestor will be inherited by descendant0 and descendant1.

As indicated above, texture mapping will only be applied to components containing rendered polygonal meshes for which appropriate texture coordinates have been set. Determining such texture coordinates that produce appropriate results for a given source image is often non-trivial; this so-called “u-v mapping problem” is difficult in general and is highly dependent on the mesh geometry. ArtiSynth users can handle the problem of assigning texture coordinates in several ways:

- Use meshes which already have appropriate texture coordinates defined for a given source image. This generally means that mesh is specified by a file that contains the required texture coordinates. The mesh should then be read from this file (Section 2.5.5) and then used in the construction of the relevant components. For example, the application can read in a mesh containing texture coordinates and then use it to create a RigidBody via the method RigidBody.createFromMesh().

- Use a simple mesh object with predefined texture coordinates. The class MeshFactory provides the methods

  ```java
  PolygonalMesh createRectangle (width, height, xdivs, ydivs, addTextureCoords);
  PolygonalMesh createSphere (radius, nslices, nlevels, addTextureCoords)
  ```

which create rectangular and spherical meshes, along with canonical canonical texture coordinates if addTextureCoords is true. Coordinates generated by createSphere() are defined so that (0,0) and (1,1) map to the spherical coordinates (−π,π) (at the south pole) and (π,0) (at the north pole). Source images can be relatively easy to find for objects with canonical coordinates.

- Compute custom texture coordinates and set them within the mesh using setTextureCoords().

An example where texture mapping is applied to spherical meshes to make them appear like tennis balls is defined in artisynth.demos.tutorial.SphericalTextureMapping and listing for this is given below:

```java
package artisynth.demos.tutorial;
import java.awt.Color;
import maspack.geometry.*;
```
**Figure 4.1:** Color and bump mapping applied to spherical meshes. Left: color mapping only. Middle: bump mapping only. Right: combined color and bump mapping.

```java
import maspack.matrix.RigidTransform3d;
import maspack.render.*;
import maspack.render.Renderer.ColorMixing;
import maspack.render.Renderer.Shading;
import maspack.util.PathFinder;
import maspack.spatialmotion.SpatialInertia;
import artisynth.core.mechmodels.*;
import artisynth.core.workspace.RootModel;

/**
 * Simple demo showing color and bump mapping applied to spheres to make them
 * look like tennis balls.
 */
public class SphericalTextureMapping extends RootModel {

    RigidBody createBall (MechModel mech, String name, PolygonalMesh mesh, double xpos) {
        double density = 500;
        RigidBody ball = RigidBody.createFromMesh(name, mesh.clone(), density, /*scale=*/1);
        ball.setPose(new RigidTransform3d(/*x,y,z=*/xpos, 0, 0));
        mech.addRigidBody(ball);
        return ball;
    }

    public void build (String[] args) {

        // create MechModel and add to RootModel
        MechModel mech = new MechModel("mech");
        addModel(mech);

        double radius = 0.0686;
        // create the balls
        PolygonalMesh mesh = MeshFactory.createSphere (radius, 20, 10, /*texture=*/true);

        RigidBody ball0 = createBall(mech, "ball0", mesh, -2.5*radius);
        RigidBody ball1 = createBall(mech, "ball1", mesh, 0);
        RigidBody ball2 = createBall(mech, "ball2", mesh, 2.5*radius);

        // set up the basic render props: no shininess, smooth shading to enable
        // bump mapping, and an underlying diffuse color of white to combine with
        // the color map
        RenderProps.setSpecular(mech, Color.BLACK);
        RenderProps.setShading(mech, Shading.SMOOTH);
    }
```
RenderProps.setFaceColor(mech, Color.WHITE);

// create and add the texture maps (provided free courtesy of www.robinwood.com).

String dataFolder = PathFinder.expand("${srcdir SphericalTextureMapping}/data");

ColorMapProps cprops = new ColorMapProps();
cprops.setEnabled(true);
// no specular coloring since ball should be matt
cprops.setSpecularColoring(false);
cprops.setFileName(dataFolder + "/TennisBallColorMap.jpg");

BumpMapProps bprops = new BumpMapProps();
bprops.setEnabled(true);
bprops.setScaling((float)radius/10);
bprops.setFileName(dataFolder + "/TennisBallBumpMap.jpg");

// apply color map to balls 0 and 2. Can do this by setting color map
// properties in the MechModel, so that properties are controlled in one
// place - but we must then also explicitly enable color mapping in
// the surface mesh components for balls 0 and 2.

RenderProps.setColorMap(mech, cprops);
RenderProps.setColorMapEnabled(ball0.getSurfaceMeshComp(), true);
RenderProps.setColorMapEnabled(ball2.getSurfaceMeshComp(), true);

// apply bump map to balls 1 and 2. Again, we do this by setting
// the render properties for their surface mesh components

RenderProps.setBumpMap(ball1.getSurfaceMeshComp(), bprops);
RenderProps.setBumpMap(ball2.getSurfaceMeshComp(), bprops);

The `build()` method uses the internal method `createBall()` to generate three rigid bodies, each defined using a spherical mesh that has been created with `MeshFactory.createSphere()` with `addTextureCoords` set to `true`. The remainder of the `build()` method sets up the render properties and the texture mappings. Two texture mappings are defined: a color mapping and bump mapping, based on the images `TennisBallColorMap.jpg` and `TennisBallBumpMap.jpg` (Figure 4.2), both located in the subdirectory `data` relative to the demo source file. `PathFinder.expand()` is used to determine the full data folder name relative to the source directory. For the bump map, it is important to set the scaling property to adjust the depth amplitude to relative to the sphere radius.

Color mapping is applied to balls 0 and 2, and bump mapping to balls 1 and 2. This is done by setting color map and/or bump map render properties in the components holding the actual meshes, which in this case is the mesh components for the balls’ surfaces meshes, obtained using `getSurfaceMeshComp()`. As mentioned above, it is also possible to set these render properties in an ancestor component, and that is done here by setting the colorMap render property of the MechModel, but then it is also necessary to enable color mapping within the individual mesh components, using `RenderProps.setColorMapEnabled()`.

To run this example in ArtiSynth, select `All demos > tutorial > SphericalTextureMapping` from the Models menu. The model should load and initially appear as in Figure 4.1. Note that if ArtiSynth is run with the legacy OpenGL 2 viewer (command line option `-GLVersion 2`), bump mapping will not be supported and will not appear.

### 4.4 Point-to-point muscles

Point-to-point muscles are a simple type of component in biomechanical models that provide muscle-activated forces acting along a line between two points. ArtiSynth provides this through `Muscle`, which is a subclass of `AxialSpring` that generates an active muscle force in response to its `excitation` property. The excitation property can be set and queried using the methods

```java
setExcitation(double a);
double getExcitation();
```
4.4.1 Muscle materials

As with AxialSprings, Muscle components use an AxialMaterial to compute the applied force $f(l, \dot{l}, a)$ in response to the muscle’s length $l$, length velocity $\dot{l}$, and excitation signal $a$. Usually the force is the sum of a passive component plus an active component that arises in response to the excitation signal.

The default AxialMaterial for a Muscle is SimpleAxialMuscle, which is essentially an activated version of LinearAxialMaterial and which computes a simple force according to

$$f(l, \dot{l}) = k(l - l_0) + d \dot{l} + m_f a$$  \hspace{1cm} (4.1)

where $k$ and $d$ are stiffness and damping terms, $a$ is the excitation value, and $m_f$ is the maximum excitation force. $k$, $d$ and $m_f$ are exposed through the properties stiffness, damping, and maxForce.

More complex muscle materials are typically used for biomechanical modeling applications, generally with nonlinear passive terms and active terms that depend on the muscle length $l$. Some of those available in ArtiSynth include ConstantAxialMuscle, BlemkerAxialMuscle, MasoudMillardLAM, and PeckAxialMuscle.

**ConstantAxialMuscle**

This is a simple muscle material that has a contractile force proportional to its activation, a constant passive tension, and linear damping, as follows:

$$f(\dot{l}) = a f_{\text{max}} + f_{\text{passive}} f_{\text{max}} + d \dot{l}$$  \hspace{1cm} (4.2)

The parameters for this material include:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_{\text{max}}$</td>
<td>maxForce</td>
<td>the maximum contractile force</td>
</tr>
<tr>
<td>$f_{\text{passive}}$</td>
<td>passiveFraction</td>
<td>the proportion of $f_{\text{max}}$ to apply as passive tension damping</td>
</tr>
<tr>
<td>$d$</td>
<td>damping</td>
<td>the Muscle’s activation</td>
</tr>
<tr>
<td>$a$</td>
<td>Muscle:excitation</td>
<td>the Muscle’s activation</td>
</tr>
</tbody>
</table>

**LinearAxialMuscle**

This is a simple muscle material that has a linear relationship between length and tension, as well as linear damping, as follows:
\[
l' = \frac{l - l_{opt}}{l_{max} - l_{opt}}, \text{ with } 0 \leq l' \leq 1 \text{ enforced} \quad (4.3)
\]

\[
f(l, \dot{l}) = a f_{max} l' + f_{passive} f_{max} l' + d \dot{l} \quad (4.4)
\]

The parameters for this material include:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(l_{max})</td>
<td>maxLength</td>
</tr>
<tr>
<td>(l_{opt})</td>
<td>optLength</td>
</tr>
<tr>
<td>(f_{max})</td>
<td>maxForce</td>
</tr>
<tr>
<td>(f_{passive})</td>
<td>passiveFraction</td>
</tr>
<tr>
<td>(d)</td>
<td>damping</td>
</tr>
<tr>
<td>(a)</td>
<td>Muscle:excitation</td>
</tr>
</tbody>
</table>

**BlemkerAxialMuscle**

This muscle material generates a force as described in [3]. It is the axial muscle equivalent to the constitutive equation along the muscle fiber direction specified in the **BlemkerMuscle** FEM material.

**PeckAxialMuscle**

This muscle material generates a force as described in [15]. It has a typical Hill-type active force-length relationship (modeled as a cosine), but the passive force-length properties are linear. This muscle model was empirically verified for jaw muscles during wide jaw opening [15].

**MasoudMillardLAM**

This muscle material generates a force as described in [12] for the rigid tendon case.

### 4.4.2 Example: Muscle attached to a rigid body

![Figure 4.3: SimpleMuscle model loaded into ArtiSynth.](image)

A simple model showing a single muscle connected to a rigid body is defined in

`artisynth.demos.tutorial.SimpleMuscle`
This model is identical to RigidBodySpring described in Section 3.2.2, except that the code to create the spring is replaced with code to create a muscle with a SimpleAxialMuscle material:

```java
// create the muscle:
muscle = new Muscle("mus", /*restLength=*/0);
muscle.setPoints (pi, mkr);
muscle.setMaterial (new SimpleAxialMuscle (/*stiffness=*/20, /*damping=*/10, /*maxf=*/10));
```

Also, so that the muscle renders differently, the rendering style for lines is set to SPINDLE using the convenience method

```java
RenderProps.setSpindleLines (muscle, 0.02, Color.RED);
```

To run this example in ArtiSynth, select All demos > tutorial > SimpleMuscle from the Models menu. The model should load and initially appear as in Figure 4.3. Running the model (Section 1.5.3) will cause the box to fall and sway under gravity. To see the effect of the excitation property, select the muscle in the viewer and then choose Edit properties ... from the right-click context menu. This will open an editing panel that allows the muscle’s properties to be adjusted interactively. Adjusting the excitation property using the adjacent slider will cause the muscle force to vary.

### 4.5 Distance Grids and Components

Distance grids, implemented by the class DistanceGrid, are currently used in ArtiSynth for both collision handling (Section 7.4.2) and spring and muscle wrapping around general surfaces (Section 8.3). A distance grid is a regular three-dimensional grid that is used to interpolate a scalar distance field and its associated normals. For collision handling and wrapping purposes, this distance function is assumed to be signed and is used to estimate the penetration depth and associated normal direction of a point within some 3D object.

A distance grid consists of \( n_x \times n_y \times n_z \) evenly spaced vertices partitioning a volume into \( r_x \times r_y \times r_z \) cuboid cells, with

\[
    r_x = n_x - 1, \quad r_y = n_y - 1, \quad r_z = n_z - 1.
\]

The grid has overall widths \( w_x, w_y, w_z \) along each axis, so that the widths of each cell are \( w_x/r_x, w_y/r_y, w_z/r_z \).

Scalar distances and normal vectors are stored at each vertex, and interpolated at points within each cell using trilinear interpolation. Distance values (although not normals) can also be interpolated quadratically over a composite quadratic cell composed of \( 2 \times 2 \times 2 \) regular cells. This provides a smoother result than trilinear interpolation and is currently used for muscle wrapping. To ensure that all points within the grid can be assigned a unique quadratic cell, the grid resolution is restricted so that \( r_x, r_y, r_z \) are always even. Distance grids are typically generated automatically from mesh data. More details on the actual functionality of distance grids is contained in the DistanceGrid API documentation.

When used within ArtiSynth, distance grids are generally contained within an encapsulating DistanceGridComp component, which maintains a mesh (or list of meshes) used to generate the grid, and exports properties for controlling its resolution, mesh fit, and visualization. For any object that implements CollidableBody (which includes RigidBody), its distance grid component can be obtained using the method

```java
DistanceGridComp getDistanceGridComp();
```

A distance grid maintains its own local coordinate system, which is related to world coordinates by a local-to-world transform that can be queried and set via the component methods setLocalToWorld() and getLocalToWorld(). For grids associated with a rigid body, the local coordinate system is synchronized with the rigid body’s coordinate system (which is queried and set via setPose() and getPose()). The grid’s axes are nominally aligned with the local coordinate system, although they can be subject to an additional orientation offset (e.g., see the description of the property fitWithOBB below).

By default, a DistanceGridComp generates its grid automatically from its mesh data, within an axis-aligned bounding volume with the resolutions \( r_x, r_y, r_z \) chosen automatically. However, in some cases it may be necessary to control the resolution explicitly. DistanceGridComp exports the following properties to control both the grid’s resolution and how it is fit around the mesh data:

- **resolution** A vector of 3 integers that specifies the resolutions \( r_x, r_y, r_z \). If any value in this vector is set \( \leq 0 \), then all values are set to zero and the maxResolution property is used to determine the grid divisions instead.
maxResolution  Sets the default maximum cell resolution that should be used when constructing the grid. This is the number of cells that should be used along the longest bounding volume axis, with the cell counts along the other axes adjusted to maintain a uniform cell size. If all three values of resolution are > 0, those will be used to specify the cell resolutions instead.

fitWithOBB  If true, offsets the orientation of the grid’s x, y, and z axes (with respect to local coordinates) to align with an oriented bounding box fit to the mesh. Otherwise, the grid axes are aligned with those of the local coordinate frame.

marginFraction  Specifies the fractional amount by which the mesh should be extended with respect to a bounding box fit around the mesh(es). The default value is 0.1.

As with all properties, these can be set either interactively (either using a custom control panel or by selecting the component and then choosing Edit properties ... from the right-click context menu), or through their set/get accessor methods. For example, resolution and maxResolution can be set and queried via the methods:

```java
void setResolution (Vector3i res)
Vector3i getResolution()

void setMaxResolution (int max)
int getMaxResolution()
```

When used for either collisions or wrapping, the distance values interpolated by a distance grid are used to determine whether a point is inside or outside some 3D object, with the inside/outside boundary being the isosurface surface corresponding to a distance value of 0. This isosurface surface (which differs depending on whether trilinear or quadratic distance interpolation is used) therefore represents the effective collision boundary, and it may be somewhat different from the mesh surface used to generate it. It may be smoother, or may have discretization artifacts, depending on both the smoothness and complexity of the mesh and the grid’s resolution (Figure 4.4). It is therefore important to be able to visualize both the trilinear and quadratic isosurfaces. DistanceGridComp provides a number of properties to control this along with other aspects of grid visualization:

renderProps  Render properties that control the colors, styles, and sizes (or widths) used to render faces, lines and points (Section 4.3). Point, line and edge properties are used for rendering grid vertices, normals, and cell edges, while face properties are used for rendering the isosurface. One can select which components are visible by zeroing appropriate render properties: zeroing pointRadius (or pointSize, if the pointStyle is POINT) disables drawing of the vertices; zeroing lineRadius (or lineWidth, if the lineStyle is LINE) disables drawing of the normals; and zeroing edgeWidth disables drawing of the cell edges. For an illustration, see Figure 4.5.

renderGrid  If set to true, causes the grid’s vertices, normals and cell edges to be rendered, using the render properties as described above.

renderRanges  Can be used to restrict which part of the distance grid is drawn. The value is a string which specifies the vertex ranges along the x, y, and z axes. In general, the grid will have $n_x \times n_y \times n_z$ vertices along the x, y, and z axes, where $n_x$, $n_y$, and $n_z$ are each one greater than the cell resolutions $r_x$, $r_y$, and $r_z$. The range string should contain three range specifications, one for each axis, where each specification is either * (all vertices), n:m (vertices in the index range n to m, inclusive), or n (vertices only at index n). A range specification of ** * ** (or ***) means draw all vertices, which is the default behavior. Other examples include:

```
** 7 **  all vertices along x and z, and those at index 7 along y
* 0 2 3*  a single vertex at indices (0, 2, 3)
"0:3 4:5 **" all vertices between 0 and 3 along x, and 4 and 5 along y
```

For an illustration, see Figure 4.5.

renderSurface  If set to true, renders the grid’s isosurface, as determined by the surfaceType property. See Figure 4.4, right.

surfaceType  Controls the interpolation used to form the isosurface rendered in response to the renderSurface property. QUADRATIC (the default) specifies a quadratic isosurface, while TRILINEAR specifies a trilinear isosurface.

surfaceDistance  Controls the level set value used to determine the isosurface. To render the isosurface used for collision handling or muscle wrapping, this value should be 0.
Figure 4.4: A mesh used to generate a distance grid, (left), along with a visualization of the grid itself (middle) and the corresponding quadratic isosurface (right). Notice how in this case the quadratic isosurface is smoother than the coarser features of the original mesh.

Figure 4.5: Rendering a subportion of a distance grid restricted along the x axis by setting renderRanges to "9:12 *, with render properties set to show the grid cells (left), and the vertices and normals (right).

When visualizing the isosurface for a distance grid, it is generally convenient to also turn off visualization for the meshes used to generate the grid. For RigidBody objects, this can be accomplished easily using the convenience property gridSurfaceRendering. If set true, it will cause the isosurface to be rendered instead of its mesh components. The isosurface type will be that indicated by the grid component’s surfaceType property, and the rendering will occur independently of the visibility settings for the meshes or the grid component.

4.6 Transforming geometry

Certain ArtiSynth components, including MechModel, implement the interface TransformableGeometry, which allows the geometric transformation of the component’s attributes (such as meshes, points, frame locations, etc.), along with its descendant components. The interface provides the method

```java
public void transformGeometry (AffineTransform3dBase X);
```

where X is an AffineTransform3dBase that may be either a RigidTransform3d or a more general AffineTransform3d (Section 2.2).
**transformGeometry(X)** can be used to translate, rotate, shear or scale components. It can be applied to an entire model or individual components. Unlike **scaleDistance()**, it actually changes the physical geometry and so may change the simulation behavior. For example, applying **transformGeometry()** to a **RigidBody** will cause the shape of its mesh to change, which will change its mass if its **inertiaMethod** property is set to **DENSITY**. Figure 4.6 shows a simplified illustration of both rigid and affine transformations being applied to a model.

![Figure 4.6: Simple illustration of a model (left) undergoing a rigid transformation (middle) and an affine transformation (right).](image)

The example below shows how to apply a transformation to a model in code. In it, a **MechModel** is first scaled by the factors 1.5, 2, and 3 along the x, y, and z axes, and then flipped upside down using a **RigidTransform3d** that rotates it by 180 degrees about the x axis:

```java
MechModel mech;
...
build mech model ...

AffineTransform3d X = new AffineTransform3d();
X.applyScaling (1.5, 2, 3);
mech.transformGeometry (X);

RigidTransform3d T =
    new RigidTransform3d (/*x,y,z=*/0, 0, 0, /*r,p,y=*/0, 0, Math.PI);
mech.transformGeometry (T);
```

### 4.6.1 Nonlinear transformations

The **TransformableGeometry** interface also supports general, nonlinear geometric transforms. This can be done using a **GeometryTransformer**, which is an abstract class for performing general transformations. To apply such a transformation to a component, one can create and initialize an appropriate subclass of **GeometryTransformer** to perform the desired transformation, and then apply it using the static **transform** method of the utility class **TransformGeometryContext**:

```java
ModelComponent comp; // component to be transformed
GeometryTransformer gtr; // transformer to do the transforming
...
Instantiate and initialize the transformer ...

TransformGeometryContext.transform (comp, gtr, /*flags=*/0);
```

At present, the following subclasses of **GeometryTransformer** are available:

**RigidTransformer**

- Implements rigid 3D transformations.
**AffineTransformer**

Implements affine 3D transformations.

**FemGeometryTransformer**

Implements a general transformation, using the deformation field induced by a finite element model.

`TransformGeometryContext` also supplies the following convenience methods to apply transformations to components or collections of components:

```java
void transform (Iterable<TransformableGeometry>, GeometryTransformer, int);
void transform (TransformableGeometry[], GeometryTransformer, int);
void transform (TransformableGeometry, AffineTransform3dBase, int);
void transform (Iterable<TransformableGeometry>, AffineTransform3dBase, int);
void transform (TransformableGeometry[], AffineTransform3dBase, int);
```

The last three of these methods create an instance of either `RigidTransformer` or `AffineTransformer` for the supplied `AffineTransform3dBase`. In fact, most `TransformableGeometry` components implement their `transformGeometry(X)` method as follows:

```java
public void transformGeometry (AffineTransform3dBase X) {
    TransformGeometryContext.transform (this, X, 0);
}
```

The `FemGeometryTransformer` class is derived from the class `DeformationTransformer`, which uses the single method `getDeformation()` to obtain deformation field information at a specified reference position:

```java
void getDeformation (Vector3d p, Matrix3d F, Vector3d r)
```

If the deformation field is described by \( x' = f(x) \), then for a given reference position \( r \) (in undeformed coordinates), this method should return the deformed position \( p = f(r) \) and the deformation gradient

\[
F \equiv \frac{\partial f}{\partial x}
\]  

(4.5)

evaluated at \( r \).

`FemGeometryTransformer` obtains \( f(x) \) and \( F \) from a `FemModel3d` (see Section 6) whose elemental rest positions enclose the components to be transformed, using the fact that a finite element model creates an implied piecewise-smooth deformation field as it deviates from its rest position. For each reference point \( r \) needed by the transformation process, `FemGeometryTransformer` finds the FEM element whose rest volume encloses \( r \), and then uses the corresponding shape function coordinates to compute \( f(x) \) and \( F \) from the element’s deformation. If the FEM model does not enclose \( r \), the nearest element is used to determine the shape function coordinates (however, this calculation becomes less accurate and meaningful the farther \( r \) is from the FEM model). Transformations based on FEM models are further illustrated in Section 4.6.2, and by Figure 4.8. Full details on ArtiSynth finite element models are given in Section 6.

Besides FEM models, there are numerous other ways to create deformation fields, such as radial basis functions, thin plate splines, etc. Some of these may be more appropriate for a particular application and can provide deformations that are globally smooth (as opposed to piecewise smooth). It should be relatively easy for an application to create its own subclass of `DeformationTransformer` to implement the deformation of choice by overriding the single `getDeformation()` method.

### 4.6.2 Example: the `FemModelDeformer` class

An FEM-based geometric transformation of a `MechModel` is facilitated by the class `FemModelDeformer`, which one can add to an existing `RootModel` to transform the geometry of a `MechModel` already located within that `RootModel`. `FemModelDeformer` subclasses `FemModel3d` to include a `FemGeometryTransformer`, and provides some utility methods to support the transformation process.

A `FemModelDeformer` can be added to a `RootModel` by adding the following code fragment to the end of the `build()` method:
public void build (String[] args) {
    ...
    build the model ...

    FemModelDeformer deformer = new FemModelDeformer ("deformer", this, /* maxn=*/10);
    addModel (deformer);
    // add a control panel (this is optional)
    addControlPanel (deformer.createControlPanel());
}

When the deformer is created, its constructor searches the specified RootModel to locate the first top-level MechModel. It then creates a hexahedral FEM grid around this model, with maxn specifying the number of cells along the maximum dimension. Material and mass properties of the model are computed automatically from the underlying MechModel dimensions (but can be altered if necessary after construction). When added to the RootModel, the deformer becomes another top-level model that can be deformed independently of the MechModel to create the required deformation field, as described below. It also supplies application-defined menu items that appear under the Application menu in the ArtiSynth menu bar (see Section 5.5). The deformer’s createControlPanel() can also be used to create a ControlPanel (Section 5.1) that controls the visibility of the FEM model and the dynamic behavior of both it and the MechModel.

An example is defined in

    artisynth.demos.tutorial.DeformedJointedCollide

where the JointedCollide example of Section 7.1.3 is extended to include a FemModelDeformer using the code described above.

![Figure 4.7: The DeformedJointedCollide example initially loaded into ArtiSynth.](image)

To load this example in ArtiSynth, select All demos > tutorial > DeformedJointedCollide from the Models menu. The model should load and initially appear as in Figure 4.7, where the control panel appears on the right.

The underlying MechModel (or "model") can now be transformed by first deforming the FEM model (or "grid") and then using the resulting deformation field to effect the transformation:

1. Make the model non-dynamic and the grid dynamic by unchecking model dynamic and checking grid dynamic in the control panel. This means that when simulation is run, the model will be inert while the grid will respond physically.
2. Deform the grid using simulation. One easy way to do this is to fix certain nodes, generally on or near the grid boundary, and then move some of these using the translation or transrotator tool while simulation is running. To fix a set of nodes, select them in the viewer, choose Edit properties ... from the right-click context menu, and then uncheck their dynamic property. To easily select a large number of nodes without also selecting model components or grid elements, one can specify FemNode in the selection filter widget. (See the sections “Transformer Tools” and “Selection filtering” in the ArtiSynth User Interface Guide.)

3. After the grid has been deformed, choose deform from the Application menu in the ArtiSynth toolbar to transform the model. Afterwards, the transformation can be undone by choosing undo, and the grid can be reset by choosing reset grid.

4. To run the deformed model after the transformation, it should again be made dynamic by checking model dynamic in the control panel. The itself grid can be made non-dynamic, and it and/or its nodes can be made invisible by unchecking grid visible and/or grid nodes visible in the control panel.

The result of a possible deformation is shown in Figure 4.8.

![Deformation achieved in DeformedJointedCollide, showing both the model and grid (using an orthographic view) before and after the deformation.](image)

**Note:** FemModelDeformer is not intended to provide a general purpose solution to nonlinear geometric transformations. Rather, it is mainly intended to illustrate the capabilities of GeometryTransformer and the TransformableGeometry interface.

### 4.6.3 Implementation and behavior

As indicated above, the management of transforming the geometry for one or more components is handled by the TransformGeometryContext class. The transform operations themselves are carried out by this class’s apply() method, which (a) assembles all the components that need to be transformed, (b) performs the actual transform operations, (c) invokes any required updating actions on other components, and finally (d) notifies parent components of the change using a GeometryChangeEvent.

To support this, ArtiSynth components which implement TransformableGeometry must also supply the methods

```java
public void addTransformableDependencies(TransformGeometryContext context, int flags);

public void transformGeometry(GeometryTransformer gtr, TransformGeometryContext context, int flags);
```

The first method, addTransformableDependencies(context,flags), is called in step (a) to add to the context any additional components which should be transformed along with this component. This includes any descendants which should be transformed, since the transformation of these should not generally be done within
transformGeometry(gtr, context, flags).

The second method, `transformGeometry(gtr, context, flags)`, is called in step (b) to perform the actual transformation on this component. It should use the supplied geometry transformer `gtr` to transform its attributes, as well as `context` to query what other components are also being transformed and to request any needed updating actions to be called in step (c). The `flags` argument specifies conditions associated with the transformation, which at the moment may currently include:

**TG_SIMULATING**

The system is currently simulating, and therefore it may not be desirable to transform all attributes;

**TG_ARTICULATED**

Rigid body articulation constraints should be enforced as the transform proceeds.

Full details for all this are given in the documentation for `TransformGeometryContext`.

The transforming behavior of a component is up to its implementing method, but the following rules are generally observed:

1. Transformable descendants are also transformed, by using `addTransformableDependencies()` to add them to the context as described above;

2. When the nodes of an FEM model (Section 6) are transformed, the rest positions are also transformed if the system is not simulating (i.e., if the `TG_SIMULATING` flag is not set). This also causes the mass of the adjacent nodes to be recomputed from the densities of the adjacent elements;

3. When dynamic components are transformed, any attachments and constraints associated with them are updated appropriately, but only if the system is not simulating. Non-transforming dynamic components that are attached to transforming components as slaves are generally updated so as to follow the transforming components to which they are attached.

### 4.6.4 Use in model registration

Transforming model geometry can obviously be used as part of the process of creating subject-specific biomechanical and anatomical models. However, registration will generally require more than geometric transformation, since other properties, such as material stiffnesses, densities, and maximum forces will generally need to be adjusted as well.

As a specific example, when applying a geometric transform to a model containing `AxialSprings`, the `restLength` properties of the springs will be unchanged, whereas the initial lengths may be, resulting in a different applied forces and physical behavior.

### 4.7 General component arrangements

As discussed in Section 1.1.5 and elsewhere, a `MechModel` provides a number of predefined child components for storing particles, rigid bodies, springs, constraints, and other components. However, applications are not required to store their components in these containers, and may instead create any sort of component arrangement desired.

For example, suppose that one wishes to create a biomechanical model of both the right and left human arms, consisting of bones, point-to-point muscles, and joints. The standard containers supplied by `MechModel` would require that all the components be placed within the following containers:

```plaintext
rigidBodies // all bones
axialSprings // all point-to-point muscles
connectors // all joints
```

Instead of this, one may wish to set up a more appropriate component hierarchy, such as
leftArm // left-arm components
bones // left bones
muscles // left muscles
joints // left joints
rightArm // right-arm components
bones // right bones
muscles // right muscles
joints // right joints

To do this, the application build() method can create the necessary hierarchy and then populate it with whatever components are desired. Before simulation begins (or whenever the model structure is changed), the MechModel will recursively traverse the component hierarchy and update whatever internal structures are needed to run the simulation.

### 4.7.1 Container components

The generic classComponentList can be used as a container for model components of a specific type. It can be created using a declaration of the form

```java
ComponentList<Particle> list = new ComponentList<Type> (Type.class, name);
```

where Type is the class type of the components and name is the name for the container. Once the container is created, it should be added to the MechModel (or another internal container) and populated with child components of the specified type. For example,

```java
MechModel mech;
...
ComponentList<Particle> parts =
    new ComponentList<Particle> (Particle.class, "parts");
ComponentList<Frame> frames =
    new ComponentList<Frame> (Frame.class, "frames");

// add containers to the mech model
mech.add (parts);
mech.add (frames);
```

creates two containers named "parts" and "frames" for storing components of type Particle and Frame, respectively, and adds them to a MechModel referenced by mech.

In addition to ComponentList, applications may use several "specialty" container types which are subclasses of ComponentList:

**RenderableComponentList**

A subclass of ComponentList, that has its own set of render properties which can be inherited by its children. This can be useful for compartmentalizing render behavior. Note that it is not necessary to store renderable components in a RenderableComponentList; components stored in a ComponentList will be rendered too.

**PointList**

A RenderableComponentList that is optimized for rendering points, and also contains its own pointDamping property that can be inherited by its children.

**PointSpringList**

A RenderableComponentList designed for storing point-based springs. It contains a material property that specifies a default axial material that can be used by its children.

**AxialSpringList**

A PointSpringList that is optimized for rendering two-point axial springs.

If necessary, it is relatively easy to define one’s own customized list by subclassing one of the other list types. One of the main reasons for doing so, as suggested above, is to supply default properties to be inherited by the list’s descendants.
A component list which declares `ModelComponent` as its type can be used to store any type of component, including other component lists. This allows the creation of arbitrary component hierarchies. Generally either `ComponentList<ModelComponent>` or `RenderableComponentList<ModelComponent>` are best suited to implement hierarchical groupings.

### 4.7.2 Example: a net formed from balls and springs

![Net Demo model loaded into ArtiSynth.](image)

A simple example showing an arrangement of balls and springs formed into a net is defined in

```python
artisynth.demos.tutorial.NetDemo
```

The `build()` method and some of the supporting definitions for this example are shown below.

```python
protected double stiffness = 1000.0; // spring stiffness
protected double damping = 10.0; // spring damping
protected double maxForce = 5000.0; // max force with excitation = 1
protected double mass = 1.0; // mass of each ball
protected double widthX = 20.0; // width of the net along x
protected double widthY = 20.0; // width of the net along y
protected int numX = 8; // num balls along x
protected int numY = 8; // num balls along y

// custom component containers
protected MechModel mech;
protected PointList<Particle> balls;
protected ComponentList<ModelComponent> springs;
protected RenderableComponentList<AxialSpring> greenSprings;
protected RenderableComponentList<AxialSpring> blueSprings;

private AxialSpring createSpring (PointList<Particle> parts, int idx0, int idx1) {
// create a "muscle" spring connecting particles indexed by 'idx0' and
// 'idx1' in the list 'parts'
Muscle spr = new Muscle (parts.get(idx0), parts.get(idx1));
spr.setMaterial (new SimpleAxialMuscle (stiffness, damping, maxForce));
return spr;
}
```
public void build (String[] args) {

    // create MechModel and add to RootModel
    mech = new MechModel ("mech");
    mech.setGravity (0, 0, -980.0);
    mech.setPointDamping (1.0);
    addModel (mech);

    int nump = (numx+1)*(numy+1); // nump = total number of balls

    // create custom containers:
    balls = new PointList<Particle> (Particle.class, "balls");
    springs = new ComponentList<ModelComponent>(ModelComponent.class,"springs");
    greenSprings = new RenderableComponentList<AxialSpring> (AxialSpring.class, "greenSprings");
    blueSprings = new RenderableComponentList<AxialSpring> (AxialSpring.class, "blueSprings");

    // create balls in a grid pattern and add to the list 'balls'
    for (int i=0; i<=numx; i++) {
        for (int j=0; j<=numy; j++) {
            double x = widthx*(-0.5+i/(double)numx);
            double y = widthy*(-0.5+j/(double)numy);
            Particle p = new Particle (mass, x, y, /*z=*/0);
            balls.add (p);
            // fix balls along the edges parallel to y
            if (i == 0 || i == numx) {
                p.setDynamic (false);
            }
        }
    }

    // connect balls by green springs parallel to y
    for (int j=0; j< numy; j++) {
        greenSprings.add {
            createSpring (balls, i*(numy+1)+j, i*(numy+1)+j+1);
        }
    }

    // connect balls by blue springs parallel to x
    for (int i=0; i< numx; i++) {
        blueSprings.add {
            createSpring (balls, i*(numy+1)+j, (i+1)*(numy+1)+j);
        }
    }

    // add containers to the mechModel
    springs.add (greenSprings);
    springs.add (blueSprings);
    mech.add (balls);
    mech.add (springs);

    // set render properties for the components
    RenderProps.setLineColor (greenSprings, new Color(0f, 0.5f, 0f));
    RenderProps.setLineColor (blueSprings, Color.BLUE);
    RenderProps.setSphericalPoints (mech, widthx/50.0, Color.RED);
    RenderProps.setCylindricalLines (mech, widthx/100.0, Color.BLUE);
}

The build() method begins by creating a MechModel in the usual way (lines 29-30). It then creates a net composed of a set of balls arranged as a uniform grid in the x-y plane, connected by a set of green colored springs running parallel to the y axis and a set of blue colored springs running parallel to the x axis. These are arranged into a component hierarchy of the form
balls
springs
greenSprings
blueSprings

using containers created at lines 37-42. The balls are then created and added to balls (lines 45-56), the springs are created and added to greenSprings and blueSprings (lines 59-71), and the containers are added to the MechModel at lines 74-77. The balls along the edges parallel to the y axis are fixed. Render properties are set at lines 80-83, with the colors for greenSprings and blueSprings being explicitly set to dark green and blue.

MechModel, along with other classes derived from ModelBase, enforces reference containment. That means that all components referenced by components within a MechModel must themselves be contained within the MechModel. This condition is checked whenever a component is added directly to a MechModel or one of its ancestors. This means that the components must be added to the MechModel in an order that ensures any referenced components are already present. For example, in the NetDemo example above, adding the particle list after the spring list would generate an error.

To run this example in ArtiSynth, select All demos > tutorial > NetDemo from the Models menu. The model should load and initially appear as in Figure 4.9. Running the model will cause the net to fall and sway under gravity. When the ArtiSynth navigation panel is opened and expanded, the component hierarchy will appear as in Figure 4.10. While the standard MechModel containers are still present, they are not displayed by default because they are empty.

4.7.3 Adding containers to other models

In addition to MechModel, application-defined containers can be added to any model that inherits from ModelBase. This includes RootModel and FemModel. However, at the present time, components added to such containers won’t do anything, other than be rendered in the viewer if they are Renderable.

4.8 Custom Joints

If desired, it is also possible for applications to create their own custom joints. This involves creating two custom classes: a coupling class that does the constraint computations, and a joint class that wraps around it and allows it to connect connectable bodies. Details on how to create these classes are given in Sections 4.8.3 and 4.8.4, after some explanation of the constraint mechanism that underlies joint operation.

This section assumes that the reader is highly familiar with spatial kinematics and dynamics.

4.8.1 Joint constraints

To create a custom joint, it is necessary to understand how joints are implemented. The basic function of a joint is to constraint the set of poses allowed by the joint transform \( T_{CD} \) that relates frame C to D. To do this, the joint imposes
restrictions on the six-dimensional spatial velocity $\hat{v}_{CD}$ that describes how frame C is moving with respect to D. This restriction is done using a set of bilateral constraints $G_k$ and (in some cases) unilateral constraints $N_l$, each of which is a $1 \times 6$ matrix that acts to restrict a single degree of freedom in $\hat{v}_{CD}$ (see Section A.5 for a review of spatial velocities and forces). Bilateral constraints take the form of an equality,

$$G_k \hat{v}_{CD} = 0,$$

while unilateral constraints take the form of an inequality:

$$N_l \hat{v}_{CD} \geq 0.$$  

These constraints are defined with respect to frame C, and their total number equals the number of DOFs that the joint removes. A joint’s main computational task is to specify these constraints. ArtiSynth then uses its own knowledge of how frames C and D are connected to bodies A and B (or ground, if there is no body B) to map the individual $G_k$ and $N_l$ onto the joint’s full bilateral and unilateral constraint matrices $G$ and $N$ (see (3.10) and (3.11)) that restrict the body velocities.

As a simple example, consider a cylindrical joint, in which C is free to rotate about and translate along the z axis of D but other motions are restricted. Letting $v$ and $\omega$ denote the translational and rotational components of $\hat{v}_{CD}$, such that

$$\hat{v}_{CD} = \begin{pmatrix} v \\ \omega \end{pmatrix},$$

we see that the constraints must enforce

$$v_z = v_y = \omega_x = \omega_y = 0.$$  

This can be accomplished using four constraints defined as follows:

$$G_0 = (1, 0, 0, 0, 0, 0)$$
$$G_1 = (0, 1, 0, 0, 0, 0)$$
$$G_2 = (0, 0, 0, 1, 0, 0)$$
$$G_3 = (0, 0, 0, 0, 1, 0).$$

Constraining velocities is a necessary but insufficient condition for constraint enforcement. Because of numerical errors, as well as the fact that constraints are often nonlinear, the joint transform $T_{CD}$ will tend to drift away from the joint restrictions as the simulation proceeds, leading to the error $T_{err}$ described at the end of Section 3.3.1. These errors are corrected during a position correction at the end of every simulation time step: the joint first projects $T_{CD}$ onto the nearest valid constraint surface to form $T_{GD}$, and $T_{err}$ is then computed from

$$T_{err} = T_{CG} = T_{GD}^{-1}T_{CD}.$$  

Because $T_{err}$ is (usually) small, we can approximate it as a twist $\hat{d}_{err}$ representing a small displacement from frame G (which lies on the constraint surface) to frame C. During the position correction, ArtiSynth adjusts the pose of C relative to D in order to try and bring $\hat{d}_{err}$ to zero. To do this, it uses an estimate of the distance $d_k$ along each constraint to the constraint surface, which it computes from the dot product of $G_k$ and $\hat{d}_{err}$:

$$d_k = G_k \hat{d}_{err}.$$  

ArtiSynth assembles these distances into a composite distance vector $d_x$ for all bilateral constraints, and then uses the system solver to find a displacement $\delta q$ of the system coordinates that satisfies

$$G(q) \delta q = -d_x.$$  

Adding $\delta q$ to the system coordinates $q$ then reduces the constraint errors. While for nonlinear constraints several steps may be required to bring the error to zero, the process usually converges quickly.

Unlike bilateral constraints, unilateral constraints are one-sided, and take effect, or are engaged, only when $T_{CD}$ encounters an inadmissible region. The constraint then acts to prevent further penetration into the region, via the velocity restriction (4.7), and also to push $T_{CD}$ out of the inadmissible region, using a position correction analogous to that used for bilateral constraints.

Whether or not a unilateral constraint is engaged is determined by its engaged value $E_l$, which takes one of the three values: $\{0, 1, -1\}$, and is updated by the joint implementation as the simulation proceeds. A value of 0 means that the constraint is not engaged, and will not be included in the joint’s unilateral constraint matrix $N_l$. Otherwise, if $E_l$ is 1
or \(-1\), then the constraint is engaged and will be included in \(N_l\), using \(N_l\) if \(E_l = 1\), or its negative \(-N_l\) if \(E_l = -1\). \(E_l\) therefore defines a sign for the constraint. General details on how unilateral constraints should be engaged or disengaged are discussed in Section 4.8.2.

A common use of unilateral constraints is to implement limits on joint coordinate values; this also illustrates the utility of \(E_l\). For example, the cylindrical joint mentioned above may have two coordinates, \(z\) and \(\theta\), describing the translation and rotation along and about the D frame’s z axis. Now suppose we wish to bound \(z\), such that

\[
 z_{\min} \leq z \leq z_{\max}. \tag{4.11}
\]

When these limits are violated, a unilateral constraint can be engaged to limit motion along the z axis. A constraint \(N_z\) that will do this is

\[
 N_z = (0, 0, 1, 0, 0, 0). \tag{4.7}
\]

Whenever \(z \leq z_{\min}\), using \(N_z\) in (4.7) will ensure that \(\dot{z} \geq 0\) and hence \(z\) will not fall further below the lower bound. On the other hand, when \(z \geq z_{\max}\), we want to employ \(-N_z\) in (4.7) to ensure that \(\dot{z} \leq 0\). In other words, lower bounds can be enforced by engaging \(N_z\) with \(E_l = 1\), while upper bounds can be enforced with \(E_l = -1\).

As with bilateral constraints, constraining velocities is not sufficient; it is also necessary to correct position errors, particularly as unilateral constraints are typically not engaged until the inadmissible region is violated. The position correction procedure is the same: for each engaged unilateral constraint, find a distance \(d_l\) along its constraint direction that indicates the distance to the inadmissible region boundary. ArtiSynth will then assemble these \(d_l\) into a composite distance vector \(d_n\) for all unilateral constraints, and solve for a system coordinate displacement \(\dot{\delta}q\) that satisfies

\[
 N(q) \delta \dot{q} \geq -d_n. \tag{4.12}
\]

Because of the inequality direction in (4.12), distances \(d_l\) representing penetration into an inadmissible region must be negative. For coordinate bounds such as (4.11), we need to use \(d_l = z - z_{\min}\) for the lower bound and \(d_l = z_{\max} - z\) for the upper bound. Alternatively, if the unilateral constraint has been included into the projection of \(C\) onto \(G\) and hence into the error term \(\dot{\delta}_{err}\), \(d_l\) can be computed from

\[
 d_l = E_l N_l \dot{\delta}_{err}. \tag{4.13}
\]

Note that unilateral constraints for coordinate limits are not usually incorporated into the \(G\) projection; more on this details are given in Section 4.8.4.

As simulation proceeds, the velocity limits imposed by (4.6) and (4.7) are enforced by bilateral and unilateral constraint forces \(f_k\) and \(f_l\) whose magnitudes are given by

\[
 f_k = G^T_k \dot{\lambda}_k, \quad f_l = N^T_l \dot{\theta}_l, \tag{4.14}
\]

where \(\dot{\lambda}_k\) and \(\dot{\theta}_l\) are the Lagrange multipliers computed by the mechanical system solver (and are components of \(\dot{\lambda}\) or \(\dot{\theta}\) in (1.8) and (1.6)). \(f_k\) and \(f_l\) are 6 DOF spatial force vectors, or wrenches (Section A.5), which like \(G_k\) and \(N_l\) are expressed in frame C. Because \(G_k^T\) and \(N_l^T\) are proportional to spatial wrenches, they are often themselves referred to as constraint wrenches, and within the ArtiSynth codebase are described by a Wrench object.

### 4.8.2 Unilateral constraint engagement

As mentioned above, joints which implement unilateral constraints must monitor \(T_{CD}\) and the joint coordinates as the simulation proceeds and decide when to engage or disengage them.

Engagement is usually easy: a constraint is engaged whenever \(T_{CD}\) or a joint coordinate hits an inadmissible region. The constraint \(N_l\) is itself a spatial vector that is (locally) perpendicular to the inadmissible region boundary, and \(E_l\) is chosen to be either 1 or \(-1\) so that \(E_l N_l\) is directed away from the inadmissible region. In the remainder of this section, we shall assume \(E_l = 1\).

To disengage, we usually want to ensure that the joint configuration is out of the inadmissible region. If we have a constraint \(N_l\), with a local distance \(d_l\) defined such that \(d_l < 0\) implies the joint is inside the region, then we are out of the region when \(d_l > 0\). However, if we use only this as the disengagement criterion, we may encounter a problem known as chattering, illustrated in Figure 4.11 (left). An inadmissible region is shown in gray, with a unilateral constraint \(N_l\) perpendicular to its boundary. As simulation proceeds, the joint lands inside the region at an initial point \(A\) at the lower left, at a (negative) distance \(d_1\) from the boundary. Ideally the position correction step will move the configuration by \(-d_1\) so that it lands right on the region boundary. However, numerical errors and nonlinearities may
ArtiSynth implements two solutions to chattering. One is to implement a deadband, so that instead of correcting the position by \(-d_1\), we correct it by \(-d_1 - p_{\text{tol}}\), where \(p_{\text{tol}}\) is a penetration tolerance. This means that the correction will try to leave the joint inside the region by a small amount (Figure 4.11, right) so that chattering is suppressed.

The penetration tolerance used depends on the constraint type. Those that are primarily linear use the value of the \(\text{penetrationTol}\) property, while those that are primarily rotary use the value of the \(\text{rotaryLimitTol}\) property; both of these are exported as inheritable properties by both \text{MechModel} and \text{BodyConnector}, with default values computed from the model’s overall dimensions.

The second chattering solution is to disengage only when the joint is actively moving away from the region, as determined by \(d_l > 0\). The disengagement criteria then become

\[
d_l > 0 \quad \text{and} \quad \dot{d}_l > 0.
\]  

(4.15)

\(\dot{d}_l\) is called the contact speed and can be computed from

\[
\dot{d}_l = N_l \hat{v}_{CD}.
\]  

(4.16)

Another problem, which we call constraint oscillation, can occur when we are near two or more overlapping inadmissible regions whose boundaries are not perpendicular. See Figure 4.12 (left), which shows two overlapping regions 1 and 2. The joint starts at point A, inside region 1 but just outside region 2. Since only constraint 1 is engaged, the position correction moves it toward the boundary of 1, overshooting and landing at point B outside of 1 but inside region 2. Constraint 2 now engages, moving the joint to C, where it is past the boundary of 2 but inside 1 again. While the example in the figure converges to the corner where the boundaries of 1 and 2 meet, convergence may be slow and may
be prevented entirely by external forcing. While the mechanisms that prevent chattering may also prevent oscillation, we find that an additional measure is useful, which is to simply require that a constraint must be engaged for at least two simulation steps. The result is shown in Figure 4.12 (right), where after the joint arrives at B, constraint 1 remains engaged along with constraint 2, and the subsequent solution takes the joint directly to point F at the corner where 1 and 2 meet.

### 4.8.3 Implementing a custom joint

All of the work of computing joint constraints and coordinates, as described in the previous sections, is done within a “coupling” class which is a subclass of `RigidBodyCoupling`. An instance of this is then embedded within a “joint” class (which is a subclass of `JointBase`) that supports connections with other bodies, provides rendering, exports various properties, and allows the joint to be attached to a `MechModel`.

For purposes of this discussion, we will assume that these two custom classes are called `CustomCoupling` and `CustomJoint`, respectively. The implementation of `CustomJoint` can be as simple as this:

```java
import artisynth.core.mechmodels.ConnectableBody;
import artisynth.core.mechmodels.JointBase;
import maspack.matrix.RigidTransform3d;

public class CustomJoint extends JointBase {
    public CustomJoint() {
        setCoupling(new CustomCoupling());
    }

    public CustomJoint(
            ConnectableBody bodyA, ConnectableBody bodyB, RigidTransform3d TDW) {
        this(); // call the default constructor
        setBodies(bodyA, bodyB, TDW);
    }
}
```

This creates an instance of `CustomCoupling` and sets it to the (inherited) `myCoupling` attribute inside the default constructor (which is where this normally should be done). Another constructor is provided which uses `setBodies()` to create a joint that is attached to two bodies with the D frame specified in world coordinates. In practice, a joint may also export some properties (such as joint coordinates), provide additional constructors, and implement rendering; one should examine the source code for some existing joints.

### 4.8.4 Implementing a custom coupling

Implementing a custom coupling constitutes most of the effort in creating a custom joint, since the coupling is responsible for maintaining the constraints $G_k$ and $N_l$ that enforce the joint behavior.

Before proceeding, we discuss the coordinate frame in which these constraints are situated. It is often convenient to describe joint constraints with respect to frame C, since rotations are frequently centered there. However, the joint transform $T_{CD}$ usually contains errors (Section 3.3.1) due to a combination of simulation error and possible joint compliance. To determine these errors, we project C onto another frame G, defined to be the nearest to C that is consistent with the bilateral (and possibly some unilateral) constraints. (This is done by the `projectToConstraints()` method, described below). The result is a joint transform $T_{GD}$ that is “error free” with respect to bilateral constraints and also consistent with the coordinates (if supported). This makes it convenient to formulate constraints with respect to frame G instead of C, and so this is the convention ArtiSynth uses. In particular, the `updateConstraints()` method, described below, uses $T_{GD}$, together with the spatial velocity $v_{GD}$ describing the motion of G with respect to C.

An actual custom coupling implementation involves subclassing `RigidBodyCoupling` and then implementing five abstract methods, the outline of which looks like this:

```java
import maspack.matrix.*;
import maspack.spatialmotion.*;
import maspack.util.*;

class CustomCoupling extends RigidBodyCoupling {
```
public CustomCoupling() {
    super();
}

// Initialize the constraints and coordinates.
public void initializeConstraints() {
    ...
}

// If coordinates are implemented, set TCD from the supplied coordinates.
public void coordinatesToTCD(RigidTransform3d TCD, VectorNd coords) {
    ...
}

// If coordinates are implemented, set their values from TCD.
public void TCDToCoordinates(VectorNd coords, RigidTransform3d TCD) {
    ...
}

// Project TCD to the nearest transform TGD admissible to the
// bilateral constraints, and maybe some unilateral constraints.
public void projectToConstraints(RigidTransform3d TGD, RigidTransform3d TCD, boolean updateCoords) {
    ...
}

// Update the constraint wrenches, and maybe the engaged
// and distance settings for some unilateral constraints.
public void updateConstraints(RigidTransform3d TGD, RigidTransform3d TCD, Twist errC, Twist velGD, boolean updateEngaged) {
    ...
}

The implementations of these methods are now described in detail.

initializeConstraints()

This method has the signature

public void initializeConstraints()

and is called in the coupling’s superclass constructor (i.e., the constructor for RigidBodyCoupling). It is responsible for initializing the coupling’s constraints and (if supported) coordinates.

Constraints are added using one of the two superclass methods:

RigidBodyConstraint addConstraint(int flags)

RigidBodyConstraint addConstraint(int flags, Wrench wrench)

Each creates a new RigidBodyConstraint and adds it to the coupling’s constraint list. flags is an or-ed combination of the following flags defined in RigidBodyConstraint:

BILATERAL
Constraint is bilateral (i.e., an equality). If BILATERAL is not specified, the constraint is considered unilateral.

ROTARY
Constraint primarily restricts rotary motion. If it is unilateral, the joint’s rotaryLimitTol property is used for its penetration tolerance.
LINEAR
Constraint primarily restricts translational motion. If it is unilateral, the joint’s penetrationTol property is used for its penetration tolerance.

CONSTANT
Constraint is constant with respect to frame G. This flag is set automatically if the constraint is created using addConstraint(flags, wrench).

LIMIT
Constraint is used to enforce limits for a coordinate. This flag is set automatically if the constraint is specified as the limit constraint for a coordinate.

The method addConstraint(flags, wrench) takes an additional Wrench argument specifying the (presumed constant) value of the constraint with respect to frame G, and sets the CONSTANT flag just described.

Coordinates are added similarly using using one of the two superclass methods:

```java
CoordinateInfo addCoordinate()
CoordinateInfo addCoordinate {
    double min, double max, int flags, RigidBodyConstraint limCon
}
```

Each creates a new CoordinateInfo object (which is an inner class of RigidBodyCoupling), and adds it to the coupling’s coordinate list. In the second method, min and max give the initial range limits, and limCon, if non-null, specifies a unilateral constraint (previously created using addConstraint) for enforcing the limits and causes that constraint’s LIMIT to be set. The argument flags is reserved for future use and should be set to 0. If not specified, the default coordinate limits are (−inf, inf).

The implementation of initializeConstraints() for a coupling that implements a hinge type joint might look like this:

```java
public void initializeConstraints () {
    addConstraint (BILATERAL|LINEAR, new Wrench (1, 0, 0, 0, 0, 0));
    addConstraint (BILATERAL|LINEAR, new Wrench (0, 1, 0, 0, 0, 0));
    addConstraint (BILATERAL|LINEAR, new Wrench (0, 0, 1, 0, 0, 0));
    addConstraint (BILATERAL|ROTARY, new Wrench (0, 0, 0, 1, 0, 0));
    addConstraint (ROTARY, new Wrench (0, 0, 0, 0, 1, 0));
    addCoordinate (-Math.PI, Math.PI, 0, getConstraint (5));
}
```

Six constraints are specified, with the sixth being a unilateral constraint that enforces the limits on the single coordinate describing the rotation angle. Each constraint and coordinate has an integer index giving the location in its list, in the order it was added. This index can be used to later retrieve the RigidBodyConstraint or CoordinateInfo object for the constraint or coordinate, using the methods getConstraint(idx) or getCoordinateInfo(idx).

Because initializeConstraints() is called in the superclass constructor, member attributes for the custom coupling will not yet be initialized when it is first called. Therefore, the method should not depend on the initial values of non-static member variables. initializeConstraints() can also be called later to rebuild the constraints if some defining setting is changed.

coordinatesToTCD()
This method has the signature

```java
public void coordinatesToTCD (RigidTransform3d TCD, VectorNd coords)
```

and is called when needed by the system. If coordinates are supported, then the transform T_{CD} should be set from the coordinate values supplied in coords, and returned in the argument TCD. Otherwise, this method should do nothing.
TCDToCoordinates()

This method has the signature

```java
public void TCDToCoordinates (VectorNd coords, RigidTransform3d TCD)
```

and is called when needed by the system. It is the inverse of `coordinatesToTCD()`: if coordinates are supported, then their values should be set from the joint transform $T_{CD}$ supplied by `TCD` and returned in `coords`. Otherwise, this method should do nothing.

When calling this method, it is assumed that `TCD` is “legal” with respect to the joint’s constraints (as defined by `projectToConstraints()`, described next). If this is not the case, then `projectToConstraints()` should be called instead.

One issue that can arise is when a coordinate represents an angle $\phi$ that has a range greater than $2\pi$. In that case, a common strategy is to compute a nominal value for $\phi$, and then add or subtract $2\pi$ from it until the resulting value is as close as possible to the `current` value for the angular coordinate. This allows the angle to wrap through its entire range.

To implement this, one can use the method

```java
double nearestAngle (double phi)
```

in the coordinate’s `CoordinateInfo` object, which finds the angle equivalent to `phi` that is nearest to the current coordinate value.

Coordinate values computed by this method should not be clipped to their ranges.

projectToConstraints()

This method has the signature

```java
public void projectToConstraints (RigidTransform3d TGD, RigidTransform3d TCD, VectorNd coords)
```

and is called when needed by the system. It is responsible for projecting the joint transform $T_{CD}$ (supplied by `TCD`) onto the nearest transform $T_{GD}$ that is valid for the bilateral constraints, and returning this in `TGD`. If coordinates are supported and `coords` is non-null, then the coordinate values corresponding to $T_{GD}$ should also be computed and returned in `coords`. The easiest way to do this is to simply call `TCDToCoordinates(TGD,coords)`, although in some cases it may be computationally cheaper to compute both the coordinates and the projection at the same time.

Optionally, the coupling may also extend the projection to include unilateral constraints that are not associated with coordinate limits. In particular, this should be done for constraints for which it is desired to have the constraint error included in $T_{err}$, and the corresponding argument $errC$ that is passed to `updateConstraints()`.

updateConstraints()

This method has the signature

```java
public void updateConstraints (RigidTransform3d TGD, RigidTransform3d TCD, Twist errC, Twist velGD, boolean updateEngaged)
```

and is usually called once per simulation time step. It is responsible for:

- Updating the values of all non-constant constraint wrenches, along with their derivatives;
- If `updateEngaged` is `true`, updating the `engaged` and `distance` attributes for all unilateral constraints not associated with a coordinate limit.

The method supplies several arguments:
• TGD, containing the idealized joint transform $T_{GD}$ from frame G to D produced by calling `projectToConstraints()`.

• TCD, containing the joint transform $T_{CD}$ from frame C to D and supplied for legacy reasons.

• errC, representing the (hopefully small) error transform $T_{err}$ from frame C to G as a spatial twist vector.

• velGD, giving the spatial velocity $\dot{v}_{GD}$ of frame G with respect to D, as seen in G; this is needed to compute wrench derivatives.

• updateEngaged, which requests the updating of unilateral engaged and distance attributes as described above.

If the coupling supports coordinates, their values will be updated before the method is called so as to correspond to $T_{GD}$. If needed, a coordinate’s value may be obtained from the value attribute of its `CoordinateInfo` object, which may in turn be obtained using `getCoordinateInfo(idx)`. Likewise, `ConstraintInfo` objects for each constraint may be obtained using `getConstraint(idx)`.

Constraint wrenches correspond to $G_k$ and $N_l$ in Section 4.8.1. These, along with their derivatives $\dot{G}_k$ and $\dot{N}_l$, are described by the `wrenchG` and `dotWrenchG` attributes of each constraint’s `RigidBodyConstraint` object, and may be managed by a variety of methods:

```csharp
Wrench getWrenchG() // return the reference to wrenchG
void setWrenchG (double fx, double fy, double fz, double mx, double my, double mz)
void setWrenchG (Vector3d f, Vector3d m) // either f or m may be null
void setWrenchG (Wrench wr)
void negateWrenchG ()
void zeroWrenchG ()

Wrench getDotWrenchG() // return the reference to dotWrenchG
void setDotWrenchG (double fx, double fy, double fz, double mx, double my, double mz)
void setDotWrenchG (Vector3d f, Vector3d m) // either f or m may be null
void setDotWrenchG (Wrench wr)
void negateDotWrenchG ()
void zeroDotWrenchG ()
```

`dotWrenchG` is used in computing the time derivative terms $g$ and $n$ that appear in (3.9) and (1.6). While these improve the computational accuracy of the simulation, their effect is often small, and so in practice one may be able to omit computing `dotWrenchG` and instead leave its value as 0.

Wrench information must also be computed for unilateral constraints which implement coordinate limits. While it is not necessary to compute the distance and engaged attributes for these constraints (this is done automatically), it is necessary to ensure that the wrench’s magnitude is compatible with the coordinate’s speed. More precisely, if the coordinate is given by $\phi$, then the limit wrench $N_l$ must have a magnitude such that

$$\dot{\phi} = N_l \dot{v}_{GD}.$$  \hspace{1cm} (4.17)

As mentioned above, if `updateEngaged` is `true`, the engaged and distance attributes for unilateral constraints not associated with coordinate limits must be updated. These correspond to $E_l$ and $d_l$ in Section 4.8.1, and are contained in the constraint’s `RigidBodyConstraint` object and may be queried using the methods

```csharp
double getDistance ()
void setDistance (double d)

int getEngaged ()
void setEngaged (int engaged)
```

It is up to `updateConstraints()` to compute the distance, with a negative value denoting penetration into the inadmissible region. If `projectToConstraints()` is implemented so as to account for the constraint, then $T_{GD}$ will be projected out of the inadmissible region and the distance will be implicitly present $T_{err}$ and so can be recovered by taking the dot product of the constraint wrench and `velGD`: 
RigidBodyConstraint cons = getConstraint (3); // assume constraint index is 3

... double dist = cons.getWrench().dot (velGD);

Otherwise, if the constraint is not accounted for in `projectToConstraints()`, the distance must be obtained by other means.

To update `engaged`, one may use the general convenience method

```java
void updateEngaged (RigidBodyConstraint cons, double dist, double dmin, double dmax, Twist velGD)
```

which sets `engaged` according to the rules of Section 4.8.2, for an inadmissible region corresponding to `dist < dmin` or `dist > dmax`. The upper or lower bounds may be removed by setting `dmin` to `-inf` or `max` to `inf`, respectively.

### 4.8.5 Example: a simple custom joint

![Coordinate frames for the CustomJoint (left) and the associated CustomJointDemo (right).](image)

An simple model illustrating custom joint creation is provided by

```java
artisynth.demos.tutorial.CustomJointDemo
```

This implements a joint class defined by `CustomJoint` (also in the package `artisynth.demos.tutorial`), which is actually just a simple implementation of `SlottedHingeJoint` (Section 3.4.4). Certain details are omitted, such as exporting coordinate values and ranges as properties, and other things are simplified, such as the rendering code. One may consult the source code for `SlottedHingeJoint` to obtain a more complete example.

This section will focus on the implementation of the joint coupling, which is created as an inner class of `CustomJoint` called `CustomCoupling` and which (like all couplings) extends `RigidBodyCoupling`. The joint itself creates an instance of the coupling in its default constructor, exactly as described in Section 4.8.3.

The coupling allows two DOFs (Figure 4.13, left): translation along the `x` axis of D (described by the coordinate `x`), and rotation about the `z` axis of D (described by the coordinate `θ`), with `T_{CD}` related to the coordinates by (3.25). It implements `initializeConstraints()` as follows:

```java
public void initializeConstraints () {
    addConstraint (BILATERAL | LINEAR);
    addConstraint (BILATERAL | LINEAR, new Wrench (0, 0, 1, 0, 0, 0));
    addConstraint (BILATERAL | ROTARY, new Wrench (0, 0, 0, 1, 0, 0));
    addConstraint (BILATERAL | ROTARY, new Wrench (0, 0, 0, 1, 0, 0));
    addConstraint (LINEAR);
    addConstraint (ROTARY, new Wrench (0, 0, 0, 0, 1, 0));
    addCoordinate (-1, 1, 0, getConstraint (4)); // x
    addCoordinate (-2*Math.PI, 2*Math.PI, 0, getConstraint (5)); // theta
}
```
Six constraints are added using `addConstraint()`: two linear bilaterals to restrict translation along the y and z axes of D, two rotary bilaterals to restrict rotation about the x and y axes of D, and two unilaterals to enforce limits on x and \( \theta \). Four of the constraints are constant in frame G, and so are initialized with a wrench value. The other two are not constant in G and so will need to be updated in `updateConstraints()`. The coordinates for x and \( \theta \) are added at the end, using `addCoordinate()` with default joint limits and a reference to the constraint that will enforce the limit.

The implementations for `coordinatesToTCD()` and `TCDToCoordinates()` simply use (3.25) to compute \( T_{CD} \) from the coordinates, or vice versa:

```java
public void coordinatesToTCD (RigidTransform3d TCD, VectorNd coords) {
    double x = coords.get(X_IDX);
    double theta = coords.get(THETA_IDX);
    TCD.setIdentity();
    TCD.p.x = x;
    double c = Math.cos(theta);
    double s = Math.sin(theta);
    TCD.R.m00 = c;
    TCD.R.m11 = c;
    TCD.R.m01 = -s;
    TCD.R.m10 = s;
}

public void TCDToCoordinates (VectorNd coords, RigidTransform3d TGD) {
    coords.set(X_IDX, TGD.p.x);
    double theta = Math.atan2(TGD.R.m10, TGD.R.m00);
    coords.set(THETA_IDX, getCoordinateInfo(THETA_IDX).nearestAngle(theta));
}
```

`X_IDX` and `THETA_IDX` are constants defining the coordinate indices for x and \( \theta \). In `TCDToCoordinates()`, note the use of the `CoordinateInfo` method `nearestAngle()`, as discussed in Section 4.8.4.

Projecting \( T_{CD} \) onto the error-free \( T_{GD} \) is done by `projectToConstraints()`, implemented as follows:

```java
public void projectToConstraints (RigidTransform3d TGD, RigidTransform3d TCD, VectorNd coords) {
    TGD.R.set(TCD.R);
    TGD.R.rotateZDirection(Vector3d.Z_UNIT);
    TGD.p.x = TCD.p.x;
    TGD.p.y = 0;
    TGD.p.z = 0;
    if (coords != null) {
        TCDToCoordinates (coords, TGD);
    }
}
```

The translational projection is easy - the y and z components of the translation vector \( p \) are simply zeroed out. To project the rotation \( R \), we use its `rotateZDirection()` method, which applies the shortest rotation aligning its z axis with (0, 0, 1). The residual rotation will be a rotation in the x-y plane. If `coords` is non-null and needs to be computed, we simply call `TCDToCoordinates()`.

Lastly, the implementation for `projectToConstraints()` is as follows:

```java
public void updateConstraints (RigidTransform3d TGD, RigidTransform3d TCD, Twist errC, Twist velGD, boolean updateEngaged) {
    RigidBodyConstraint cons = getConstraint(0); // constraint along y
    double s = TGD.R.m10; // sin (theta)
    double c = TGD.R.m00; // cos (theta)
    // constraint wrench along y is constant in D but needs to be transformed to G
    cons.setWrenchG (s, c, 0, 0, 0, 0);
    // derivative term:
    double dotTheta = velGD.w.z;
    cons.setDotWrenchG (c*dotTheta, -s*dotTheta, 0, 0, 0, 0);
    // update x limit constraint if necessary
```
Only constraints 0 and 4 need to have their wrenches updated, since the rest are constant, and we obtain their constraint objects using `getConstraint(idx)`. Constraint 0 restricts motion along the y axis in D, and while this is constant in D, it is not constant in G, which is where the wrench must be situated. The y axis of D as seen in G is given by the second row of the rotation matrix of \( T_{GD} \), which from (3.25) we see is \((s, c, 0)^T\), where \( s \equiv \sin(\theta) \) and \( c \equiv \cos(\theta) \). We obtain \( s \) and \( c \) directly from \( T_{GD} \), since this has been projected to lie on the constraint surface; alternatively, we could compute them from \( \theta \). To obtain the wrench derivative, we note that \( \dot{s} = c \dot{\theta} \) and \( \dot{c} = -s \dot{\theta} \), and that \( \dot{\theta} \) is simply the z component of the angular velocity of G with respect to D, or velGD.w.z. The wrench and its derivative are set using the constraint’s `setWrenchG()` and `setDotWrenchG()` methods.

The other non-constant constraint is the limit constraint for the x coordinate, which is the x axis of D as seen in G. This is updated similarly, although we only need to do so if the limit constraint is engaged. Since all unilateral constraints are coordinate limits, there is no need to update their `distance` or `engaged` attributes as this is done automatically by the system.
Chapter 5

Simulation Control

This section describes different devices which an application may use to control the simulation. These include control panels to allow for the interactive adjustment of properties, as well as agents which are applied every time step. Agents include controllers and input probes to supply and modify input parameters at the beginning of each time step, and monitors and output probes to observe and record simulation results at the end of each time step.

5.1 Control Panels

A control panel is an editing panel that allows for the interactive adjustment of component properties.

It is always possible to adjust component properties through the GUI by selecting one or more components and then choosing Edit properties ... in the right-click context menu. However, it may be tedious to repeatedly select the required components, and the resulting panels present the user with all properties common to the selection. A control panel allows an application to provide a customized editing panel for selected properties.

5.1.1 General principles

Control panels are implemented by the ControlPanel model component. They can be set up within a model’s build() method by creating an instance of ControlPanel, populating it with widgets for editing the desired properties, and then adding it to the root model using the latter’s addControlPanel() method.

One of the most commonly used means of adding widgets to a control panel is the method addWidget(comp,propertyPath), which creates a widget for a property specified by propertyPath with respect to the component comp. Property paths are discussed in the Section 1.4.2, and can consist solely of a property name, or, for properties located in descendant components, a component path followed by a colon ‘:’ and the property name.

Other flavors of addWidget() also exist, as described in the API documentation for ControlPanel. In addition to property widgets, any type of Swing or awt component can be added using the method addWidget(awtcomp).

Control panels can also be created interactively using the GUI; see the section “Control Panels” in the ArtiSynth User Interface Guide.

5.1.2 Example: Creating a simple control panel

An application model showing a control panel is defined in

    artisynth.demos.tutorial.SimpleMuscleWithPanel

This model simply extends SimpleMuscle (Section 4.4.2) to provide a control panel for adjusting gravity, the mass and color of the box, and the muscle excitation. The class definition, excluding include statements, is shown below:
The `build()` method calls `super.build()` to create the model used by `SimpleMuscle`. It then proceeds to create a `ControlPanel`, populate it with widgets, and add it to the root model (lines 8-15). The panel is given the name "controls" in the constructor (line 8); this is its component name and is also used as the title for the panel’s window frame. A control panel does not need to be named, but if it is, then that name must be unique among the control panels.

Lines 9-11 create widgets for three properties located relative to the `MechModel` referenced by `mech`. The first is the `MechModel`’s gravity. The second is the mass of the box, which is a component located relative to `mech` by the path name (Section 1.1.3) "rigidBodies/box". The third is the box’s face color, which is the sub-property `faceColor` of the box’s `renderProps` property.

Line 12 adds a `JSeparator` to the panel, using the `addWidget()` method that accepts general components, and line 13 adds a widget to control the `excitation` property for `muscle`.

It should be noted that there are different ways to specify target properties in `addWidget()`. First, component paths may contain numbers instead of names, and so the box’s mass property could be specified using "rigidBodies/0:mass" instead of "rigidBodies/box:mass" since the box’s number is 0. Second, if a reference to a subcomponent is available, one can specify properties directly with respect to that, instead of indicating the subcomponent in the property path. For example, if the box was referenced by a variable `body`, then one could use the construction

```java
panel.addWidget (body, "mass");
```

in place of

```java
panel.addWidget (mech, "rigidBodies/box:mass");
```

To run this example in ArtiSynth, select All demos > tutorial > SimpleMuscleWithPanel from the Models menu. The properties shown in the panel can be adjusted interactively by the user, while the model is either stationary or running.
5.2 Custom properties

Because of the usefulness of properties in creating control panels and probes (Sections 5.1 and Section 5.4), model developers may wish to add their own properties, either to the root model, or to a custom component.

This section provides only a brief summary of how to define properties. Full details are available in the “Properties” section of the Maspack Reference Manual.

5.2.1 Adding properties to a component

As mentioned in Section 1.4, properties are class-specific, and are exported by a class through code contained in the class’s definition. Often, it is convenient to add properties to the RootModel subclass that defines the application model. In more advanced applications, developers may want to add properties to a custom component.

The property definition steps are:

Declare the property list:

The class exporting the properties creates its own static instance of a PropertyList, using a declaration like

```java
static PropertyList myProps = new PropertyList (MyClass.class, MyParent.←
class);
@Override
public PropertyList getAllPropertyInfo () {  
    return myProps;
}
```

where MyClass and MyParent specify the class types of the exporting class and its parent class. The PropertyList declaration creates a new property list, with a copy of all the properties contained in the parent class. If one does not want the parent class properties, or if the parent class does not have properties, then one would use the constructor PropertyList(MyClass.class) instead. If the parent class is an ArtiSynth model component (including the RootModel), then it will always have its own properties. The declaration of the method getAllPropertyInfo() exposes the property list to other classes.

Add properties to the list:

Properties can then be added to the property list, by calling the PropertyList’s add() method:

```java
PropertyDesc add (String name, String description, Object defaultValue);
```

where name contains the name of the property, description is a comment describing the property, and defaultValue is an object containing the property’s default value. This is done inside a static code block:

```java
static {
    myProps.add ("stiffness", "spring stiffness", /*defaultValue=*/1);
    myProps.add ("damping", "spring damping", /*defaultValue=*/0);
}
```

Variations on the add() method exist for adding read-only or inheritable properties, or for setting various property options. Other methods allow the property list to be edited.

Declare property accessor functions:

For each property propXXX added to the property list, accessor methods of the form

```java
void setPropXXX (TypeX value) {
    ...  
}
TypeX getPropXXX() {
    TypeX value = ...
    return value;
}
```
must be declared, where TypeX is the value associated with the property.

It is possible to specify different names for the accessor functions in the string argument name supplied to the add() method. Read-only properties only require a get accessor.

5.2.2 Example: a visibility property

An model illustrating the exporting of properties is defined in

    artisynth.demos.tutorial.SimpleMuscleWithProperties

This model extends SimpleMuscleWithPanel (Section 4.4.2) to provide a custom property boxVisible that is added to the control panel. The class definition, excluding include statements, is shown below:

```java
public class SimpleMuscleWithProperties extends SimpleMuscleWithPanel {

    // internal property list; inherits properties from SimpleMuscleWithPanel
    static PropertyList myProps =
        new PropertyList (
            SimpleMuscleWithProperties.class, SimpleMuscleWithPanel.class);

    // override getAllPropertyInfo() to return property list for this class
    public PropertyList getAllPropertyInfo () {
        return myProps;
    }

    // add new properties to the list
    static {
        myProps.add (*boxVisible*, *box is visible*, false);
    }

    // declare property accessors
    public boolean getBoxVisible () {
        return box.getRenderProps().isVisible();
    }

    public void setBoxVisible (boolean visible) {
        RenderProps.setVisible (box, visible);
    }

    public void build (String[] args) throws IOException {
        super.build (args);

        panel.addWidget (this, *boxVisible*);
        panel.pack();
    }
}
```

First, a property list is created for the application class SimpleMuscleWithProperties.class which contains a copy of all the properties from the parent class SimpleMuscleWithPanel.class (lines 4-6). This property list is made visible by overriding getAllPropertyInfo() (lines 9-11). The boxVisible property itself is then added to the property list (line 15), and accessor functions for it are declared (lines 19-25).

The build() method calls super.build() to perform all the model creation required by the super class, and then adds an additional widget for the boxVisible property to the control panel.

To run this example in ArtiSynth, select All demos > tutorial > SimpleMuscleWithProperties from the Models menu. The control panel will now contain an additional widget for the property boxVisible as shown in Figure 5.2. Toggling this property will make the box visible or invisible in the viewer.
5.3 Controllers and monitors

Application models can define custom controllers and monitors to control input values and monitor output values as a simulation progresses. Controllers are called every time step immediately before the `advance()` method, and monitors are called immediately after (Section 1.1.4). An example of controller usage is provided by ArtiSynth’s inverse modeling feature, which uses an internal controller to estimate the actuation signals required to follow a specified motion trajectory.

More precise details about controllers and monitors and how they interact with model advancement are given in the ArtiSynth Reference Manual.

5.3.1 Implementation

Applications may declare whatever controllers or monitors they require and then add them to the root model using the methods `addController()` and `addMonitor()`. They can be any type of `ModelComponent` that implements the `Controller` or `Monitor` interfaces. For convenience, most applications simply subclass the default implementations `ControllerBase` or `MonitorBase` and then override the necessary methods.

The primary methods associated with both controllers and monitors are:

```java
public void initialize (double t0);
public void apply (double t0, double t1);
public boolean isActive();
```

`apply(t0, t1)` is the “business” method and is called once per time step, with `t0` and `t1` indicating the start and end times `t0` and `t1` associated with the step. `initialize(t0)` is called whenever an application model’s state is set (or reset) at a particular time `t0`. This occurs when a simulation is first started or after it is reset (with `t0 = 0`), and also when the state is reset at a waypoint or during adaptive stepping.

`isActive()` controls whether a controller or monitor is active; if `isActive()` returns `false` then the `apply()` method will not be called. The default implementations `ControllerBase` and `MonitorBase`, via their superclass `ModelAgentBase`, also provide a `setActive()` method to control this setting, and export it as the property `active`. This allows controller and monitor activity to be controlled at run time.

To enable or disable a controller or monitor at run time, locate it in the navigation panel (under the RootModel’s controllers or monitors list), chose Edit properties ... from the right-click context menu, and set the active property as desired.

Controllers and monitors may be associated with a particular model (among the list of models owned by the root model). This model may be set or queried using

```java
void setModel (Model m);
Model getModel();
```

If associated with a model, `apply()` will be called immediately before (for controllers) or after (for monitors) the model’s `advance()` method. If not associated with a model, then `apply()` will be called before or after the advance of all the models owned by the root model.
Controllers and monitors may also contain state, in which case they should implement the relevant methods from the HasState interface.

Typical actions for a controller include setting input forces or excitation values on components, or specifying the motion trajectory of parametric components (Section 3.1.3). Typical actions for a monitor include observing or recording the motion profiles or constraint forces that arise from the simulation.

When setting the position and/or velocity of a dynamic component that has been set to be parametric (Section 3.1.3), a controller should not set its position or velocity directly, but should instead set its target position and/or target velocity, since this allows the solver to properly interpolate the position and velocity during the time step. The methods to set or query target positions and velocities for Point-based components are

```java
setTargetPosition(Point3d pos);
Point3d getTargetPosition (); // read-only
setTargetVelocity(Vector3d vel);
Vector3d getTargetVelocity (); // read-only
```

while for Frame-based components they are

```java
setTargetPosition(Point3d pos);
setTargetOrientation(AxisAngle axisAng);
setTargetPose(RigidTransform3d TFW);
Point3d getTargetPosition (); // read-only
AxisAngle getTargetOrientation (); // read-only
RigidTransform3d getTargetPose (); // read-only

setTargetVelocity(Twist vel);
Twist getTargetVelocity (); // read-only
```

### 5.3.2 Example: A controller to move a point

A model showing an application-defined controller is defined in

```java
artisynth.demos.tutorial.SimpleMuscleWithController
```

This simply extends SimpleMuscle (Section 4.4.2) and adds a controller which moves the fixed particle p1 along a circular path. The complete class definition is shown below:

```java
package artisynth.demos.tutorial;

import java.io.IOException;
import maspack.matrix.*;
import artisynth.core.modelbase.*;
import artisynth.core.mechmodels.*;
import artisynth.core.gui.*;

public class SimpleMuscleWithController extends SimpleMuscleWithPanel {

    private class PointMover extends ControllerBase {

        Point myPnt; // point to be moved
        Point3d myPos0; // initial point position

        public PointMover (Point pnt) {
            myPnt = pnt;
            myPos0 = new Point3d (pnt.getPosition());
        }

        public void apply (double t0, double t1) {
            double ang = Math.PI*t1/2; // angle associated with time t1
            Point3d pos = new Point3d (myPos0);
        }
    }
}
```
A controller called `PointMover` is defined by extending `ControllerBase` and overriding the `apply()` method. It stores the point to be moved in `myPnt`, and the initial position in `myPos0`. The `apply()` method computes a target position for the point that starts at `myPos0` and then moves in a circle in the `x−z` plane with an angular velocity of \( \frac{\pi}{2} \) rad/sec (lines 22-28).

The `build()` method calls `super.build()` to create the model used by `SimpleMuscle`, and then creates an instance of `PointMover` to move particle `p1` and adds it to the root model (line 34). The viewer bounds are updated to make the circular motion more visible (line 36).

To run this example in ArtiSynth, select `All demos > tutorial > SimpleMuscleWithController` from the `Models` menu. When the model is run, the fixed particle `p1` will trace out a circular path in the `x−z` plane.

### 5.4 Probes

In addition to controllers and monitors, applications can also attach streams of data, known as `probes`, to input and output values associated with the simulation. Probes derive from the same base class `ModelAgentBase` as controllers and monitors, but differ in that

1. They are associated with an explicit time interval during which they are applied;
2. They can have an attached file for supplying input data or recording output data;
3. They are displayable in the ArtiSynth `timeline` panel.

A probe is applied (by calling its `apply()` method) only for time steps that fall within its time interval. This interval can be set and queried using the following methods:

```java
setStartTime (double t0);
setStopTime (double t1);
setInterval (double t0, double t1);

double getStartTime();
double getStopTime();
```

The probe’s attached file can be set and queried using:

```java
setAttachedFileName (String fileName);
String getAttachedFileName ();
```

where `fileName` is a string giving the file’s path name.

Details about the timeline display can be found in the section “The Timeline” in the ArtiSynth User Interface Guide.

There are two types of probe: `input probes`, which are applied at the beginning of each simulation step before the controllers, and `output probes`, which are applied at the end of the step after the monitors.
While applications are free to construct any type of probe by subclassing either \texttt{InputProbe} or \texttt{OutputProbe}, most applications utilize either \texttt{NumericInputProbe} or \texttt{NumericOutputProbe}, which explicitly implement streams of numeric data which are connected to properties of various model components. The remainder of this section will focus on numeric probes.

As with controllers and monitors, probes also implement a \texttt{isActive()} method that indicates whether or not the probe is active. Probes that are not active are not invoked. Probes also provide a \texttt{setActive()} method to control this setting, and export it as the property \texttt{active}. This allows probe activity to be controlled at run time.

\begin{tcolorbox}[colback=ht, colframe=ht]
To enable or disable a probe at run time, locate it in the navigation panel (under the RootModel’s \texttt{inputProbes} or \texttt{outputProbes} list), chose \texttt{Edit properties ...} from the right-click context menu, and set the \texttt{active} property as desired. Probes can also be enabled or disabled in the timeline, by either selecting the probe and invoking \texttt{activate} or \texttt{deactivate} from the right-click context menu, or by clicking the track mute button (which activates or deactivates all probes on that track).
\end{tcolorbox}

\subsection{5.4.1 Numeric probe structure}

Numeric probes are associated with:

\begin{itemize}
\item A vector of temporally-interpolated numeric data;
\item One or more properties to which the probe is bound and which are either set by the numeric data (input probes), or used to set the numeric data (output probes).
\end{itemize}

The numeric data is implemented internally by a \texttt{NumericList}, which stores the data as a series of vector-valued knot points at prescribed times \( t_k \) and then interpolates the data for an arbitrary time \( t \) using an interpolation scheme provided by \texttt{Interpolation}.

Some of the numeric probe methods associated with the interpolated data include:

\begin{verbatim}
int getVsize(); // returns the size of the data vector
setInterpolationOrder (Order order); // sets the interpolation scheme
Order getInterpolationOrder(); // returns the interpolation scheme
VectorNd getData (double t); // interpolates data for time t
NumericList getNumericList(); // returns the underlying NumericList
\end{verbatim}

Interpolation schemes are described by the enumerated type \texttt{Interpolation.Order} and presently include:

\begin{description}
\item[Step] Values at time \( t \) are set to the values of the closest knot point \( k \) such that \( t_k \leq t \).
\item[Linear] Values at time \( t \) are set by linear interpolation of the knot points \((k,k+1)\) such that \( t_k \leq t \leq t_{k+1} \).
\item[Parabolic] Values at time \( t \) are set by quadratic interpolation of the knots \((k-1,k,k+1)\) such that \( t_k \leq t \leq t_{k+1} \).
\item[Cubic] Values at time \( t \) are set by cubic Catmull interpolation of the knots \((k-1,\ldots,k+2)\) such that \( t_k \leq t \leq t_{k+1} \).
\end{description}

Each property bound to a numeric probe must have a value that can be mapped onto a scalar or vector value. Such properties are know as numeric properties, and whether or not a value is numeric can be tested using \texttt{NumericConverter.isNumeric(value)}.

By default, the total number of scalar and vector values associated with all the properties should equal the size of the interpolated vector (as returned by \texttt{getVsize()}). However, it is possible to establish more complex mappings between the property values and the interpolated vector. These mappings are beyond the scope of this document, but are discussed in the sections “General input probes” and “General output probes” of the \texttt{ArtiSynth User Interface Guide}.
5.4.2 Creating probes in code

This section discusses how to create numeric probes in code. They can also be created and added to a model graphically, as described in the section “Adding and Editing Numeric Probes” in the ArtiSynth User Interface Guide.

Numeric probes have a number of constructors and methods that make it relatively easy to create instances of them in code. For NumericInputProbe, there is the constructor

```
NumericInputProbe (ModelComponent c, String propPath, String filePath);
```

which creates a NumericInputProbe, binds it to a property located relative to the component `c` by `propPath`, and then attaches it to the file indicated by `filePath` and loads data from this file (see Section 5.4.4). The probe’s start and stop times are specified in the file, and its vector size is set to match the size of the scalar or vector value associated with the property.

To create a probe attached to multiple properties, one may use the constructor

```
NumericInputProbe (ModelComponent c, String propPaths[], String filePath);
```

which binds the probe to multiple properties specified relative to `c` by `propPaths`. The probe’s vector size is set to the sum of the sizes of the scalar or vector values associated with these properties.

For NumericOutputProbe, one may use the constructor

```
NumericOutputProbe (ModelComponent c, String propPath, String filePath, double ← sample);
```

which creates a NumericOutputProbe, binds it to the property `propPath` located relative to `c`, and then attaches it to the file indicated by `filePath`. The argument `sample` indicates the `sample time` associated with the probe, in seconds; a value of 0.01 means that data will be added to the probe every 0.01 seconds. If `sample` is specified as -1, then the sample time will default to the maximum step size associated with the root model.

To create an output probe attached to multiple properties, one may use the constructor

```
NumericOutputProbe (ModelComponent c, String propPaths[], String filePath, double sample);
```

As the simulation proceeds, an output probe will accumulate data, but this data will not be saved to any attached file until the probe’s `save()` method is called. This can be requested in the GUI for all probes by clicking on the `Save` button in the timeline toolbar, or for specific probes by selecting them in the navigation panel (or the timeline) and then choosing `Save data` in the right-click context menu.

Output probes created with the above constructors have a default interval of [0, 1]. A different interval may be set using `setInterval()`, `setStartTime()`, or `setStopTime()`.

5.4.3 Example: probes connected to SimpleMuscle

A model showing a simple application of probes is defined in

`artisynth.demos.tutorial.SimpleMuscleWithProbes`

This extends SimpleMuscle (Section 4.4.2) to add an input probe to move particle `p1` along a defined path, along with an output probe to record the velocity of the frame marker. The complete class definition is shown below:

```
package artisynth.demos.tutorial;
import java.io.IOException;
import maspack.matrix.*;
import maspack.util.PathFinder;
import artisynth.core.modelbase.*;
```
The input and output probes are added using the custom methods `createInputProbe()` and `createOutputProbe()`. At line 14, `createInputProbe()` creates a new input probe bound to the `targetPosition` property for the component `particles/p1` located relative to the `MechModel mech`. The same constructor attaches the probe to the file `simpleMuscleP1Pos.txt`, which is read to load the probe data. The format of this and other probe data files is described in Section 5.4.4. The method `PathFinder.getSourceRelativePath()` is used to locate the file relative to the source directory for the application model (see Section 2.6). The probe is then given the name "Particle Position" (line 18) and added to the root model (line 19).

Similarly, `createOutputProbe()` creates a new output probe which is bound to the `velocity` property for the component `particles/0` located relative to `mech`, is attached to the file `simpleMuscleMkrVel.txt` located in the application model source directory, and is assigned a sample time of 0.01 seconds. This probe is then named "FrameMarker Velocity" and added to the root model.

The `build()` method calls `super.build()` to create everything required for `SimpleMuscle`, calls `createInputProbe()` and `createOutputProbe()` to add the probes, and adjusts the MechModel viewer bounds to make the resulting probe motion more visible.

To run this example in ArtiSynth, select `All demos > tutorial > SimpleMuscleWithProbes` from the Models menu. After the model is loaded, the input and output probes should appear on the timeline (Figure 5.3). Expanding the probes should display their numeric contents, with the knot points for the input probe clearly visible. Running the model will cause particle `p1` to trace the trajectory specified by the input probe, while the velocity of the marker is recorded in the output probe. Figure 5.4 shows an expanded view of both probes after the simulation has run for about six seconds.

### 5.4.4 Data file format

The data files associated with numeric probes are ASCII files containing two lines of header information followed by a set of knot points, one per line, defining the numeric data. The time value (relative to the probe’s start time) for each knot point can be specified explicitly at the start of the each line, in which case the file takes the following format:
Figure 5.3: Timeline view of the probes created by SimpleMuscleWithProbes.

Figure 5.4: Expanded view of the probes after SimpleMuscleWithProbes has run for about 6 seconds, showing the data accumulated in the output probe "FrameMarker Velocity".

```
startTime stopTime scale
interpolation vsize explicit
t0 val00 val01 val02 ...
t1 val10 val11 val12 ...
t2 val20 val21 val22 ...
...
```

Knot point information begins on line 3, with each line being a sequence of numbers giving the knot’s time followed by \( n \) values, where \( n \) is the vector size of the probe (i.e., the value returned by \( \text{getVsize()} \)).

Alternatively, time values can be implicitly specified starting at 0 (relative to the probe’s start time) and incrementing by a uniform timeStep, in which case the file assumes a second format:

```
startTime stopTime scale
interpolation vsize timeStep
val00 val01 val02 ...
val10 val11 val12 ...
val20 val21 val22 ...
...
```

For both formats, startTime, stopTime, and scale are numbers giving the probe’s start and stop time in seconds and scale gives the scale factor (which is typically 1.0). interpolation is a word describing how the data should be interpolated between knot points and is the string value of Interpolation.Order as described in Section 5.4.1 (and which is typically Linear, Parabolic, or Cubic). vsize is an integer giving the probe’s vector size.

The last entry on the second line is either a number specifying a (uniform) time step for the knot points, in which case the file assumes the second format, or the keyword explicit, in which case the file assumes the first format.
As an example, the file used to specify data for the input probe in the example of Section 5.4.3 looks like the following:

```
0 4.0 1.0
Linear 3 explicit
0.0 0.0 0.0 0.0
1.0 0.5 0.0 0.5
2.0 0.0 0.0 1.0
3.0 -0.5 0.0 0.5
4.0 0.0 0.0 0.0
```

Since the data is uniformly spaced beginning at 0, it would also be possible to specify this using the second file format:

```
0 4.0 1.0
Linear 3 1.0
0.0 0.0 0.0
0.5 0.0 0.5
0.0 0.0 1.0
-0.5 0.0 0.5
0.0 0.0 0.0
```

### 5.4.5 Adding probe data in-line

It is also possible to specify input probe data directly in code, instead of reading it from a file. For this, one would use the constructor

```
NumericInputProbe (ModelComponent c, String propPath, double t0, double t1);
```

which creates a `NumericInputProbe` with the specified property and with start and stop times indicated by `t0` and `t1`. Data can then be added to this probe using the method

```
addData (double[] data, double timeStep);
```

where `data` is an array of knot point data. This contains the same knot point information as provided by a file (Section 5.4.4), arranged in row-major order. Times values for the knots are either implicitly specified, starting at 0 (relative to the probe’s start time) and increasing uniformly by the amount specified by `timeStep`, or are explicitly specified at the beginning of each knot if `timeStep` is set to the built-in constant `NumericInputProbe.EXPLICIT_TIME`. The size of the `data` array should then be either \( n \times m \) (implicit time values) or \( (n + 1) \times m \) (explicit time values), where \( n \) is the probe’s vector size and \( m \) is the number of knots.

As an example, the data for the input probe in Section 5.4.3 could have been specified using the following code:

```
NumericInputProbe p1probe =
    new NumericInputProbe (mech, "particles/pl:targetPosition", 0, 5);
p1probe.addData (new double[] {
    0.0, 0.0, 0.0, 0.0,
    1.0, 0.5, 0.0, 0.5,
    2.0, 0.0, 0.0, 1.0,
    3.0, -0.5, 0.0, 0.5,
    4.0, 0.0, 0.0, 0.0 }, NumericInputProbe.EXPLICIT_TIME);
```

When specifying data in code, the interpolation defaults to `Linear` unless explicitly specified using `setInterpolationOrder()`, as in, for example:

```
probe.setInterpolationOrder (Order.Cubic);
```
5.4.6 Numeric monitor probes

In some cases, it may be useful for an application to deploy an output probe in which the data, instead of being collected from various component properties, is generated by a function within the probe itself. This ability is provided by a NumericMonitorProbe, which generates data using its `generateData(vec, t, trel)` method. This evaluates a vector-valued function of time at either the absolute time `t` or the probe-relative time `trel` and stores the result in the vector `vec`, whose size equals the vector size of the probe (as returned by `getVsize()`). The probe-relative time `trel` is determined by

\[
\text{trel} = (t - \text{tstart}) / \text{scale} \quad (5.1)
\]

where `tstart` and `scale` are the probe’s start time and scale factors as returned by `getStartTime()` and `getScale()`.

As described further below, applications have several ways to control how a NumericMonitorProbe creates data:

- Provide the probe with a DataFunction using the `setDataFunction(func)` method;
- Override the `generateData(vec, t, trel)` method;
- Override the `apply(t)` method.

The application is free to generate data in any desired way, and so in this sense a NumericMonitorProbe can be used similarly to a Monitor, with one of the main differences being that the data generated by a NumericMonitorProbe can be automatically displayed in the ArtiSynth GUI or written to a file.

The DataFunction interface declares an `eval()` method,

```java
void eval (VectorXd vec, double t, double trel)
```

that for NumericMonitorProbes evaluates a vector-valued function of time, where the arguments take the same role as for the monitor’s `generateData()` method. Applications can declare an appropriate DataFunction and set or query it within the probe using the methods

```java
void setDataFunction (DataFunction func);
DataFunction getDataFunction();
```

The default implementation `generateData()` checks to see if a data function has been specified, and if so, uses that to generate the probe data. Otherwise, if the probe’s data function is null, the data is simply set to zero.

To create a NumericMonitorProbe using a supplied DataFunction, an application will create a generic probe instance, using one of its constructors such as

```java
NumericMonitorProbe (vsize, fileName, startTime, stopTime, interval);
```

and then define and instantiate a DataFunction and pass it to the probe using `setDataFunction()`. It is not necessary to supply a file name (i.e., `fileName` can be null), but if one is provided, then the probe’s data can be saved to that file. A complete example of this is defined in

```java
artisynth.demos.tutorial.SinCosMonitorProbe
```

the listing for which is:

```java
package artisynth.demos.tutorial;

import maspack.matrix.*;
import maspack.util.Clonable;

import artisynth.core.workspace.RootModel;
import artisynth.core.probes.NumericMonitorProbe;
import artisynth.core.probes.DataFunction;

/**
 * Simple demo using a NumericMonitorProbe to generate sine and cosine waves.
 */
```
public class SinCosMonitorProbe extends RootModel {

    // Define the DataFunction that generates a sine and a cosine wave
    class SinCosFunction implements DataFunction, Cloneable {
        public void eval (VectorNd vec, double t, double trel) {
            // vec should have size == 2, one for each wave
            vec.set (0, Math.sin (t));
            vec.set (1, Math.cos (t));
        }
    }

    public void build (String[] args) {
        // Create a NumericMonitorProbe with size 2, file name "sinCos.dat", start
        // time 0, stop time 10, and a sample interval of 0.01 seconds:
        NumericMonitorProbe sinCosProbe = new NumericMonitorProbe (2, "sinCos.dat", 0, 10, 0.01);

        // then set the data function:
        sinCosProbe.setDataFunction (new SinCosFunction ());
        addOutputProbe (sinCosProbe);
    }
}

In this example, the DataFunction is implemented using the class SinCosFunction, which also implements Cloneable and the associated clone() method. This means that the resulting probe will also be duplicatable within the GUI. Alternatively, one could implement SinCosFunction by extending DataFunctionBase, which implements Cloneable by default. Probes containing DataFunctions which are not Cloneable will not be duplicatable.

When the example is run, the resulting probe output is shown in the timeline image of Figure 5.5.

Figure 5.5: Output from a NumericMonitorProbe which generates sine and cosine waves.

As an alternative to supplying a DataFunction to a generic NumericMonitorProbe, an application can instead subclass NumericMonitorProbe and override either its generateData(vec, t, trel) or apply(t) methods. As an example of the former, one could create a subclass as follows:

public class SinCosProbe extends NumericMonitorProbe {

    public SinCosProbe (String fileName, double startTime, double stopTime, double interval) {
        super (2, fileName, startTime, stopTime, interval);
    }

    public void generateData (VectorNd vec, double t, double trel) {
        vec.set (0, Math.sin (t));
        vec.set (1, Math.cos (t));
    }
}
Note that when subclassing, one must also create constructor(s) for that subclass. Also, NumericMonitorProbes which don’t have a DataFunction set are considered to be clonable by default, which means that the clone() method may also need to be overridden if cloning requires any special handling.

### 5.4.7 Numeric control probes

In other cases, it may be useful for an application to deploy an input probe which takes numeric data, and instead of using it to modify various component properties, instead calls an internal method to directly modify the simulation in any way desired. This ability is provided by a NumericControlProbe, which applies its numeric data using its applyData(vec,t,treln) method. This receives the numeric input data via the vector vec and uses it to modify the simulation for either the absolute time t or probe-relative time treln. The size of vec equals the vector size of the probe (as returned by getVsize()), and the probe-relative time treln is determined as described in Section 5.4.6.

A NumericControlProbe is the Controller equivalent of a NumericMonitorProbe, as described in Section 5.4.6. Applications have several ways to control how they apply their data:

- Provide the probe with a DataFunction using the setDataFunction(func) method;
- Override the applyData(vec,t,treln) method;
- Override the apply(t) method.

The application is free to apply data in any desired way, and so in this sense a NumericControlProbe can be used similarly to a Controller, with one of the main differences being that the numeric data used can be automatically displayed in the ArtiSynth GUI or read from a file.

The DataFunction interface declares an eval() method,

```java
void eval(VectorNd vec, double t, double treln)
```

that for NumericControlProbes applies the numeric data, where the arguments take the same role as for the monitor’s applyData() method. Applications can declare an appropriate DataFunction and set or query it within the probe using the methods

```java
void setDataFunction (DataFunction func);
DataFunction getDataFunction ()
```

The default implementation applyData() checks to see if a data function has been specified, and if so, uses that to apply the probe data. Otherwise, if the probe’s data function is null, the data is simply ignored and the probe does nothing.

To create a NumericControlProbe using a supplied DataFunction, an application will create a generic probe instance, using one of its constructors such as

```java
NumericControlProbe (vsize, data, startTime, stopTime, timeStep);
NumericControlProbe (fileName);
```

and then define and instantiate a DataFunction and pass it to the probe using setDataFunction(). The latter constructor creates the probe and reads in both the data and timing information from the specified file.

A complete example of this is defined in `artisynth.demos.tutorial.SpinControlProbe`

the listing for which is:
package artisynth.demos.tutorial;

import maspack.matrix.RigidTransform3d;
import maspack.matrix.VectorNd;
import maspack.util.Clonable;
import maspack.interpolation.Interpolation;
import artisynth.core.mechmodels.MechModel;
import artisynth.core.mechmodels.RigidBody;
import artisynth.core.mechmodels.Frame;
import artisynth.core.workspace.RootModel;
import artisynth.core.probes.NumericControlProbe;
import artisynth.core.probes.DataFunction;

/**
 * Simple demo using a NumericControlProbe to spin a Frame about the z axis.
 */
public class SpinControlProbe extends RootModel {

    // Define the DataFunction that spins the body
    class SpinFunction implements DataFunction, Clonable {
        Frame myFrame;
        RigidTransform3d myTFW0; // initial frame to world transform

        SpinFunction (Frame frame) {
            myFrame = frame;
            myTFW0 = new RigidTransform3d (frame.getPose());
        }

        public void eval (VectorNd vec, double t, double trel) {
            // vec should have size == 1, giving the current spin angle
            double ang = Math.toRadians (vec.get(0));
            RigidTransform3d TFW = new RigidTransform3d ();
            TFW.R.mulRpy (ang, 0, 0);
            myFrame.setPose (TFW);
        }

        public Object clone() throws CloneNotSupportedException {
            return super.clone();
        }
    }

    public void build (String[] args) {
        MechModel mech = new MechModel (*mech*);
        addModel (mech);

        // Create a parametrically controlled rigid body to spin:
        RigidBody body = RigidBody.createBox (*box*, 1.0, 1.0, 0.5, 1000.0);
        mech.addRigidBody (body);
        body.setDynamic (*false*);

        // Create a NumericControlProbe with size 1, initial spin data
        // with time step 2.0, start time 0, and stop time 8.
        NumericControlProbe spinProbe =
            new NumericControlProbe {
                /*vsize=*/1,
                new double[] { 0.0, 90.0, 0.0, -90.0, 0.0 },
                2.0, 0.0, 8.0 );
            // set cubic interpolation for a smoother result
            spinProbe.setInterpolationOrder (Interpolation.Order.Cubic);
            // then set the data function:
        }
This example creates a simple box and then uses a `NumericControlProbe` to spin it about the z axis, using a `DataFunction` implementation called `SpinFunction`. A clone method is also implemented to ensure that the probe will be duplicatable in the GUI, as described in Section 5.4.6. A single channel of data is used to control the orientation angle of the box about z, as shown in Figure 5.6.

![SpinControlDemo](image)

Figure 5.6: Screen shot of the SpinControlDemo, showing the numeric data in the timeline.

Alternatively, an application can subclass `NumericControlProbe` and override either its `applyData(vec,trel)` or `apply(t)` methods, as described for `NumericMonitorProbes` (Section 5.4.6).

### 5.5 Application-Defined Menu Items

Application models can define custom *menu items* that appear under the Application menu in the main ArtiSynth menu bar.

This can be done by implementing the interface `HasMenuItems` in either the `RootModel` or any of its top-level components (e.g., models, controllers, probes, etc.). The interface contains a single method

```java
public boolean getMenuItems(List<Object> items);
```

which, if the component has menu items to add, should append them to `items` and return `true`. 
The RootModel and all models derived from ModelBase implement HasMenuItems by default, but with getMenuItems() returning false. Models wishing to add menu items should override this default declaration. Other component types, such as controllers, need to explicitly implement HasMenuItems.

Note: the Application menu will only appear if getMenuItems() returns true for either the RootModel or one or more of its top-level components.

getMenuItems() will be called each time the Application menu is selected, so the menu itself is created on demand and can be varied to suite the current system state. In general, it should return items that are capable of being displayed inside a Swing JMenu; other items will be ignored. The most typical item is a Swing JMenuItem. The convenience method createMenuItem(listener,text,toolTip) can be used to quickly create menu items, as in the following code segment:

```java
public boolean getMenuItems(List<Object> items) {
    items.add(GuiUtils.createMenuItem(this, "reset", "");
    items.add(GuiUtils.createMenuItem(this, "add sphere", "");
    items.add(GuiUtils.createMenuItem(this, "show flow", "");
    return true;
}
```

This creates three menu items, each with this specified as an ActionListener and no tool-tip text, and appends them to items. They will then appear under the Application menu as shown in Figure 5.7.

To actually execute the menu commands, the items returned by getMenuItems() need to be associated with an ActionListener (defined in java.awt.event), which supplies the method actionPerformed() which is called when the menu item is selected. Typically the ActionListener is the component implementing HasMenuItems, as was assumed in the example declaration of getMenuItems() shown above. RootModel and other models derived from ModelBase implement ActionListener by default, with an empty declaration of actionPerformed() that should be overridden as required. A declaration of actionPerformed() capable of handling the menu example above might look like this:

```java
public void actionPerformed(ActionEvent event) {
    String cmd = event.getActionCommand();
    if (cmd.equals("reset")) {
        resetModel();
    } else if (cmd.equals("add sphere")) {
        addSphere();
    } else if (cmd.equals("show flow")) {
        showFlow();
    }
}
```
Chapter 6

Finite Element Models

This chapter details how to construct three-dimensional finite element models, and how to couple them with the other simulation components described in previous sections (e.g. particles and rigid bodies). Finite element muscles, which have additional properties that allow them to contract given activation signals, are discussed in Section 6.9. An example FEM model of the masseter, coupled to a rigid jaw and maxilla, is shown in Figure 6.1.

![Finite element model of the masseter, coupled to the jaw and maxilla.](image)

Figure 6.1: Finite element model of the masseter, coupled to the jaw and maxilla.

6.1 Overview

The finite element method (FEM) is a numerical technique used for solving a system of partial differential equations (PDEs) over some domain. The general approach is to divide the domain into a set of building blocks, referred to as elements. These partition the space, and form local domains over which the system of equations can be locally approximated. The corners of these elements, the nodes, become control points in a discretized system. The solution is then assumed to be smoothly interpolated across the elements based on values determined at the nodes. Using this discretization, the differential system is converted into an algebraic one, which is often linearized and solved iteratively.

In ArtiSynth, the PDEs considered are the governing equations of continuum mechanics: the conservation of mass, momentum, and energy. To complete the system, a constitutive equation is required that describes the stress-strain response of the material. This constitutive equation is what distinguishes between material types. The domain is the three-dimensional space that the model occupies. This must be divided into small elements which accurately represent the geometry. Within each element, the PDEs are sampled at a set of points, referred to as integration points, and terms are numerically integrated to form an algebraic system to solve.

The purpose of the rest of this chapter is to describe the construction and use of finite elements models within ArtiSynth. It does not further discuss the mathematical framework or theory. For an in-depth coverage of the nonlinear finite element method, as applied to continuum mechanics, the reader is referred to the textbook by Bonet and Wood [4].
6.1.1 FemModel3d

The basic type of finite element model is implemented in the class FemModel3d. This class controls some properties that are used by the model as a whole. The key ones that affect simulation dynamics are:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>density</td>
<td>The density of the model</td>
</tr>
<tr>
<td>material</td>
<td>An object that describes the material’s constitutive law (i.e. its stress-strain relationship).</td>
</tr>
<tr>
<td>particleDamping</td>
<td>Proportional damping associated with the particle-like motion of the FEM nodes.</td>
</tr>
<tr>
<td>stiffnessDamping</td>
<td>Proportional damping associated with the system’s stiffness term.</td>
</tr>
</tbody>
</table>

These properties can be set and retrieved using the methods:

```java
setDensity (double density); // sets the density
double getDensity (); // gets the density

setMaterial (FemMaterial mat); // sets the FEM’s material
FemMaterial getMaterial (); // gets the FEM’s material

setParticleDamping (double d); // sets the particle (mass) damping coefficient
double getParticleDamping (); // gets the particle (mass) damping coefficient

setStiffnessDamping (double d); // sets the stiffness damping coefficient
double getStiffnessDamping (); // gets the stiffness damping coefficient
```

Keep in mind that ArtiSynth is essentially “unitless” (Section 4.2), so it is the responsibility of the developer to ensure that all properties are specified in a compatible way.

The density of the model is used to compute the mass distribution throughout the volume. Note that we use a diagonally lumped mass matrix (DLMM) formulation, so the mass is assumed to be concentrated at the location of the discretized FEM nodes. To allow for a spatially-varying density, densities can be explicitly set for individual elements, or masses can be explicitly set for individual nodes.

The FEM’s material property is a delegate object used to compute stress and stiffness within individual elements. It handles the constitutive component of the model, as described in more detail in Section 6.1.3. In addition to the main material defined for the model, it is also possible set a material on a per-element basis, and to define additional materials which augment the behavior of the main materials (Section 6.8).

The two damping parameters are related to Rayleigh damping, which is used to dissipate energy within finite element models. There are two proportional damping terms: one related to the system’s mass, and one related to stiffness. The resulting damping force applied is

\[ f_d = -(d_M M + d_K K) v, \]  

where \( d_M \) is the value of particleDamping, \( d_K \) is the value of stiffnessDamping, \( M \) is the FEM model’s lumped mass matrix, \( K \) is the FEM’s stiffness matrix, and \( v \) is the concatenated vector of FEM node velocities. Since the lumped mass matrix is diagonal, the mass-related component of damping can be applied separately to each FEM node. Thus, the mass component reduces to the same system as Equation (3.3), which is why it is referred to as “particle damping”.

6.1.2 Component Structure

Each FemModel3d contains several lists of subcomponents:

- **nodes**
  - The particle-like dynamic components of the model. These lie at the corners of the elements and carry all the mass (due to DLMM formulation).

- **elements**
  - The volumetric model elements. These define the 3D sub-units over which the system is numerically integrated.

- **shellElements**
  - The shell elements. These define additional 2D sub-units over which the system is numerically integrated.
meshes
The geometry in the model. This includes the surface mesh, and any other embedded geometries.

materials
Optional additional materials which can be added to the model to augment the behavior of the model’s material property. This is described in more detail in Section 6.8.

fields
Optional field components which can be used to interpolate application-defined quantities over the FEM model’s domain. Fields are described in detail in Section 6.10.

The nodes, elements and meshes components are illustrated in Figure 6.2.

![Figure 6.2: Subcomponents of FemModel3d.](image)

6.1.2.1 Nodes
The set of nodes belong to a finite element model can be obtained by the method

```java
PointList<FemNode3d> getNodes();  // returns list of FEM nodes
```

Nodes are implemented in the class FemNode3d, which is a subclass of Particle (Section 3.1). They are the main dynamic components of the finite element model. The key properties affecting simulation dynamics are:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>restPosition</td>
<td>The initial position of the node.</td>
</tr>
<tr>
<td>position</td>
<td>The current position of the node.</td>
</tr>
<tr>
<td>velocity</td>
<td>The current velocity of the node.</td>
</tr>
<tr>
<td>mass</td>
<td>The mass of the node.</td>
</tr>
<tr>
<td>dynamic</td>
<td>Whether the node is considered dynamic or parametric (e.g. boundary condition).</td>
</tr>
</tbody>
</table>

Each of these properties has corresponding `getXxx()` and `setXxx(...)` functions to access and modify them.

The `restPosition` property defines the node’s position in the FEM model’s “natural” or “undeformed” state. Rest positions are used to compute an initial configuration for the model, from which strains are determined. A node’s rest position can be updated in code using the method: `FemNode3d.setRestPosition(Point3d)`.

If any node’s rest positions are changed, the current values for stress and stiffness will become invalid. They can be manually updated using the method `FemModel3d.updateStressAndStiffness()` for the parent model. Otherwise, stress and stiffness will be automatically updated at the beginning of the next time step.

The properties `position` and `velocity` give the node’s current 3D state. These are common to all point-like particles, as is the `mass` property. Here, however, `mass` represents the lumped mass of the immediately surrounding material. Its
value is initialized by equally dividing mass contributions from each adjacent element, given their densities. For a finer control of spatially-varying density, node masses can be set manually after FEM creation.

The FEM node’s dynamic property specifies whether or not the node is considered when computing the dynamics of the system. If not, it is treated as being parametrically controlled. This has implications when setting boundary conditions (Section 6.1.4).

### 6.1.2.2 Elements

Elements are the 3D volumetric spatial building blocks of the domain. Within each element, the displacement (or strain) field is interpolated from displacements at nodes:

\[ u(x) = \sum_{i=1}^{N} \phi_i(x) u_i, \quad (6.2) \]

where \( u_i \) is the displacement of the \( i \)th node that is adjacent to the element, and \( \phi_i(\cdot) \) is referred to as the shape function (or basis function) associated with that node. Elements are classified by their shape, number of nodes, and shape function order (Table 6.1). ArtiSynth supports the following element types:

![Element Types](image)

**Table 6.1: Supported element types**

<table>
<thead>
<tr>
<th>Element Type</th>
<th># Nodes</th>
<th>Order</th>
<th># Integration Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>TetElement, QuadtetElement</td>
<td>4</td>
<td>linear</td>
<td>1</td>
</tr>
<tr>
<td>PyramidElement, QuadpyramidElement</td>
<td>5</td>
<td>linear</td>
<td>5</td>
</tr>
<tr>
<td>WedgeElement, QuadwedgeElement</td>
<td>6</td>
<td>linear</td>
<td>6</td>
</tr>
<tr>
<td>HexElement, QuadhexElement</td>
<td>8</td>
<td>linear</td>
<td>8</td>
</tr>
<tr>
<td>QuadtetElement</td>
<td>10</td>
<td>quadratic</td>
<td>4</td>
</tr>
<tr>
<td>QuadpyramidElement</td>
<td>13</td>
<td>quadratic</td>
<td>5</td>
</tr>
<tr>
<td>QuadwedgeElement</td>
<td>15</td>
<td>quadratic</td>
<td>9</td>
</tr>
<tr>
<td>QuadhexElement</td>
<td>20</td>
<td>quadratic</td>
<td>14</td>
</tr>
</tbody>
</table>

The base class for all of these is **FemElement3d**. A numerical integration is performed within each element to create the (tangent) stiffness matrix. This integration is performed by evaluating the stress and stiffness at a set of integration points within each element, and applying numerical quadrature. The list of elements in a model can be obtained with the method

```java
RenderableComponentList<FemElement3d> getElements();
```

All objects of type **FemElement3d** have the following properties:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>density</td>
<td>Density of the element</td>
</tr>
<tr>
<td>material</td>
<td>An object that describes the constitutive law within the element (i.e. its stress-strain relationship).</td>
</tr>
</tbody>
</table>

If left unspecified, the element’s density is inherited from the containing **FemModel3d** object. When set, the mass of the element is computed and divided amongst all its nodes, updating the lumped mass matrix.

Each element’s material property is also inherited by default from the containing **FemModel3d**. Specifying a material here allows for spatially-varying material properties across the model. Materials will be discussed further in Section 6.1.3.
6.1.2.3 Shell elements

Shell elements are additional 2D spatial building blocks which can be added to a model. They are typically used to model structures which are too thin to be easily represented by 3D volumetric elements, or to provide additional internal stiffness within a set of volumetric elements.

ArtiSynth presently supports the following shell element types, with the number of nodes, shape function order, and integration point count described in Table 6.2:

<table>
<thead>
<tr>
<th>Element Type</th>
<th># Nodes</th>
<th>Order</th>
<th># Integration Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>ShellTriElement</td>
<td>3</td>
<td>linear</td>
<td>9 (3 if membrane)</td>
</tr>
<tr>
<td>ShellQuadElement</td>
<td>4</td>
<td>linear</td>
<td>8 (4 if membrane)</td>
</tr>
</tbody>
</table>

The base class for all shell elements is ShellElement3d, which contains the same density and material properties as FemElement3d, as well as the additional property defaultThickness, whose use will be described below.

The list of shell elements in a model can be obtained with the method

```
RenderableComponentList<ShellElement3d> getShellElements();
```

Both the volumetric elements (FemElement3d) and the shell elements (ShellElement3d) derive from the base class FemElement3dBase. To obtain all the elements in an FEM model, both shell and volumetric, one may use the method

```
ArrayList<FemElement3dBase> getAllElements();
```

Each shell element can actually be instantiated in two forms:

- As a regular shell element, which has a bending stiffness;
- As a membrane element, which does not have bending stiffness.

Regular shell elements are implemented using the same extensible director formulation used by FEBio [11], and more specifically the front/back node formulation [10]. Each node associated with a (regular) shell element is assigned a director, which is a 3D vector providing a normal direction and virtual thickness at that node (Figure 6.3). This virtual thickness allows us to continue to use 3D materials to provide the constitutive laws that determine the shell’s stress/strain response, including its bending behavior. It also allows us to continue to use the element’s density to determine its mass.

Director information is automatically assigned to a FemNode3d whenever one or more regular shell elements is connected to it. This information includes both the current value of the director, its rest value, and its velocity, with the difference between the first two determining the element’s bending strain. These quantities can be queried using the methods

```
Vector3d getDirector();  // return the current director value
void getDirector (Vector3d dir);
Vector3d getRestDirector();  // return the rest director value
void getRestDirector (Vector3d dir);
Vector3d getDirectorVel();  // return the director velocity
boolean hasDirector();  // does this node have a director?
```
Figure 6.3: ShellQuadElement with the directors (dark blue lines) visible at the nodes.

For nodes which are not connected to regular shell elements, and therefore do not have director information assigned, these methods all return a zero-valued vector.

If not otherwise specified, the current and rest director values are computed automatically from the surrounding (regular) shell elements, with their value \( \mathbf{d} \) being computed from

\[
\mathbf{d} = \sum t_i \mathbf{n}_i
\]

where \( t_i \) is the value of the defaultThickness property and \( \mathbf{n}_i \) is the surface normal of the \( i \)-th surrounding regular shell element. However, if necessary, it is also possible to explicitly assign these values, using the methods

```cpp
setDirector (Vector3d dir); // set the current director
setRestDirector (Vector3d dir); // set the rest director
```

ArtiSynth FEM nodes can currently support only one director, which is shared by all regular shell elements associated with that node. This effectively means that all such elements must belong to the same “surface”, and that two intersecting surfaces cannot share the same nodes.

As indicated above, shell elements can also be instantiated as membrane elements, which do not exhibit bending stiffness and therefore do not require director information. The regular/membrane distinction is specified in the element’s constructor. For example, ShellTriElement and ShellQuadElement each have constructors with the signatures:

```cpp
ShellTriElement (FemNode3d n0, FemNode3d n1, FemNode3d n2,
                double thickness, boolean membrane);
ShellQuadElement (FemNode3d n0, FemNode3d n1, FemNode3d n2, FemNode3d n3,
                 double thickness, boolean membrane);
```

The thickness argument specifies the defaultThickness property, while membrane determines whether or not the element is a membrane element.

While membrane elements do not require explicit director information stored at the nodes, they do make use of an inferred director that is parallel to the element’s surface normal, and has a constant length equal to the element’s defaultThickness property. This gives the element a virtual volume, which (as with regular elements) is used to determine 3D strains and to compute the element’s mass from it’s density.

6.1.2.4 Meshes

The geometry associated with a finite element model consists of a collection of meshes (e.g. PolygonalMesh, PolylineMesh, PointMesh) that move along with the model in a way that maintains the shape function interpolation
equation (6.2) at each vertex location. These geometries can be used for visualizations, or for physical interactions like collisions. However, they have no physical properties themselves. FEM geometries will be discussed in more detail in Section 6.3. The list of meshes can be obtained with the method

```java
MeshComponentList<FemMeshComp> getMeshComps();
```

### 6.1.3 Materials

The stress-strain relationship within each element is defined by a “material” delegate object, implemented by a subclass of `FemMaterial`. This material object is responsible for implementing the functions

```java
void computeStressAndTangent(...)
```

which computes the stress tensor and (optionally) the tangent stiffness matrix at each integration point, based on the current local deformation at that point.

The default material type is `LinearMaterial`, where stress is related to strain through:

\[
\sigma(x) = D \varepsilon(x),
\]

where

\[
D = \begin{bmatrix}
    \lambda + 2\mu & \lambda & 0 & 0 & 0 \\
    \lambda & \lambda + 2\mu & \lambda & 0 & 0 \\
    0 & \lambda & \lambda + 2\mu & 0 & 0 \\
    0 & 0 & 0 & \mu & 0 \\
    0 & 0 & 0 & 0 & \mu \\
\end{bmatrix}
\]

\[
\lambda = \frac{E \nu}{(1 + \nu)(1 - 2\nu)}, \quad \mu = \frac{E}{2(1 + \nu)},
\]

\(\sigma\) is the standard \(6 \times 1\) stress vector, \(\varepsilon\) is the strain vector, \(E\) is the Young’s Modulus, and \(\nu\) is Poisson’s ratio. This linear material uses a corotational formulation, so rotations are removed per element before computing the strain \([14]\). To enable or disable this corotational formulation, use `LinearMaterial.setCorotated(boolean)`.

All material models, including linear and nonlinear, are available in the package `artisyth.core.materials`. A list of common materials is provided in Table 6.3. Those that are subclasses of `IncompressibleMaterial` allow for incompressibility.

### 6.1.4 Boundary conditions

Boundary conditions can be implemented in one of several ways:

1. Explicitly setting FEM node positions/velocities
2. Attaching FEM nodes to other dynamic components
3. Enabling collisions

To enforce an explicit (Dirichlet) boundary condition for a set of nodes, their `dynamic` property must be set to `false`. This notifies ArtiSynth that the state of these nodes (both position and velocity) will be controlled parametrically. By disabling dynamics, a fixed boundary condition is applied. For a moving boundary, positions and velocities of the boundary nodes must be explicitly set every timestep. This can be accomplished with either a `Controller` (see Section 5.3) or an `InputProbe` (see Section 5.4). Note that both the position and velocity of the nodes should be explicitly set for consistency.

Another type of supported boundary condition is to attach FEM nodes to other components, including particles, springs, rigid bodies, and locations within other FEM elements. Here, the node is still considered dynamic, but its motion is coupled to that of the attached component through a constraint mechanism. Attachments will be discussed further in Section 6.4.

Finally, the boundary of an FEM can be constrained by enabling collisions with other components. This will be covered in Chapter 7.

6.2 FEM model creation

Creating a finite element model in ArtiSynth typically follows the pattern:

```java
// Create and add main MechModel
MechModel mech = new MechModel("mech");
addModel(mech);

// Create FEM
FemModel3d fem = new FemModel3d("fem");

/* ... Setup FEM structure and properties ... */

// Add FEM to model
mech.addModel(fem);
```

The main code block for the FEM setup should do the following:

- Build the node/element structure
- Set physical properties
  - density
  - damping
  - material
- Set boundary conditions
- Set render properties

Building the FEM structure can be done with the use of factory methods for simple shapes, by loading external files, or by writing code to manually assemble the nodes and elements.

6.2.1 Factory methods

For simple shapes such as beams and ellipsoids, there are factory methods to automatically build the node and element structure. These methods are found in the `FemFactory` class. Some common methods are

```java
FemFactory.createGrid(...) // basic beam
FemFactory.createCylinder(...) // cylinder
FemFactory.createTube(...) // hollowed cylinder
FemFactory.createEllipsoid(...) // ellipsoid
FemFactory.createTorus(...) // torus
```

The inputs specify the dimensions, resolution, and potentially the type of element to use. The following code creates a basic beam made up of hexahedral elements:
// Create FEM
FemModel3d beam = new FemModel3d("beam");

// Build FEM structure
double[] size = {1.0, 0.25, 0.25}; // widths
int[] res = {8, 2, 2}; // resolution (# elements)
FemFactory.createGrid(beam, FemElementType.Hex,
size[0], size[1], size[2],
res[0], res[1], res[2]);
/* ... Set FEM properties ... */
// Add FEM to model
mech.addModel(beam);

### 6.2.2 Loading external FEM meshes

For more complex geometries, volumetric meshes can be loaded from external files. A list of supported file types is provided in Table 6.4. To load a geometry, an appropriate file reader must be created. Readers capable of reading FEM models implement the interface `FemReader`, which has the method

```java
readFem( FemModel3d fem ) // populates the FEM based on file contents
```

Additionally, many `FemReader` classes have static methods to handle the loading of files for convenience.

<table>
<thead>
<tr>
<th>Format</th>
<th>File extensions</th>
<th>Reader</th>
<th>Writer</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS</td>
<td>.node, .elem</td>
<td>AnsysReader</td>
<td>AnsysWriter</td>
</tr>
<tr>
<td>TetGen</td>
<td>.node, .ele</td>
<td>TetGenReader</td>
<td>TetGenWriter</td>
</tr>
<tr>
<td>Abaqus</td>
<td>.inp</td>
<td>AbaqusReader</td>
<td>AbaqusWriter</td>
</tr>
<tr>
<td>VTK (ASCII)</td>
<td>.vtk</td>
<td>VtkAsciiReader</td>
<td></td>
</tr>
</tbody>
</table>

The following code snippet demonstrates how to load a model using the `AnsysReader`.

```java
// Create FEM
FemModel3d tongue = new FemModel3d("tongue");

// Read FEM from file
try {
  // Get files relative to THIS class
  String nodeFileName = PathFinder.getSourceRelativePath(this,
    "data/tongue.node");
  String elemFileName = PathFinder.getSourceRelativePath(this,
    "data/tongue.elem");

  AnsysReader.read(tongue, nodeFileName, elemFileName);
}
``` catch (IOException ioe) {
  // Wrap error, fail to create model
  throw new RuntimeException("Failed to read model", ioe);
}

// Add to model
mech.addModel(tongue);

The method `PathFinder.getSourceRelativePath()` is used to find a path within the ArtiSynth source tree that is relative to the current model’s source file (Section 2.6). Note the try-catch block. Most of these readers throw an `IOException` if the read fails.
6.2.3 Generating from surfaces

There are two ways an FEM model can be generated from a surface: by using a FEM mesh generator, and by extruding a surface along its normal direction.

ArtiSynth has the ability to interface directly with the TetGen library (http://tetgen.org) to create a tetrahedral volumetric mesh given a closed and manifold surface. The main Java class for calling TetGen directly is TetgenTessellator. The tessellator has several advanced options, allowing for the computation of convex hulls, and for adding points to a volumetric mesh. For simply creating an FEM from a surface, there is a convenience routine within FemFactory that handles both mesh generation and constructing a FemModel3d:

```
// Create an FEM from a manifold mesh with a given quality
FemFactory.createFromMesh( PolygonalMesh mesh, double quality );
```

If quality > 0, then points will be added in an attempt to bound the maximum radius-edge ratio (see the -q switch for TetGen). According to the TetGen documentation, the algorithm usually succeeds for a quality ratio of 1.2.

It’s also possible to create thin layer of elements by extruding a surface along its normal direction.

```
// Create an FEM by extruding a surface
FemFactory.createExtrusion( FemModel3d model, int nLayers, double layerThickness, double zOffset, PolygonalMesh surface);
```

For example, to create a two-layer slice of elements centered about a surface of a tendon mesh, one might use

```
// Load the tendon surface mesh
PolygonalMesh tendonSurface = new PolygonalMesh("tendon.obj");

// Create the tendon
FemModel3d tendon = new FemModel3d("tendon");
int layers = 2; // 2 layers
double thickness = 0.0005; // 0.5 mm layer thickness
double offset = thickness; // center the layers about the surface

// Create the extrusion
FemFactory.createExtrusion( tendon, layers, thickness, offset, tendonSurface );
```

For this type of extrusion, triangular faces become wedge elements, and quadrilateral faces become hexahedral elements.

Note: for extrusions, no care is taken to ensure element quality; if the surface has a high curvature relative to the total extrusion thickness, then some elements will be inverted.

6.2.4 Building elements in code

A finite element model’s structure can also be manually constructed in code. FemModel3d has the methods:

```
addNode ( FemNode3d ); // add a node to the model
addElement ( FemElement3d ) // add an element to the model
```

For an element to successfully be added, all its nodes must already have been added to the model. Nodes can be constructed from a 3D location, and elements from an array of nodes. A convenience routine is available in FemElement3d that automatically creates the appropriate element type given the number of nodes (Table 6.1):

```
// Creates an element using the supplied nodes
FemElement3d FemElement3d.createElement( FemNode3d[] nodes );
```

Be aware of node orderings when supplying nodes. For linear elements, ArtiSynth uses a clockwise convention with respect to the outward normal for the first face, followed by the opposite node(s). To determine the correct ordering for a particular element, check the coordinates returned by the function FemElement3dBase.getNodeCoords(). This returns the concatenated coordinate list for an “ideal” element of the given type.
6.2.5 Example: a simple beam model

A complete application model that implements a simple FEM beam is given below.

```java
package artisynth.demos.tutorial;

import java.awt.Color;
import java.io.IOException;

import maspack.render.RenderProps;
import artisynth.core.femmodels.FemFactory;
import artisynth.core.femmodels.FemModel.SurfaceRender;
import artisynth.core.femmodels.FemModel3d;
import artisynth.core.femmodels.FemNode3d;
import artisynth.core.materials.LinearMaterial;
import artisynth.core.mechmodels.MechModel;
import artisynth.core.workspace.RootModel;

public class FemBeam extends RootModel {

// Models and dimensions
    FemModel3d fem;
    MechModel mech;
    double length = 1;
    double density = 10;
    double width = 0.3;
    double EPS = 1e-15;

    public void build(String[] args) throws IOException {

        // Create and add MechModel
        mech = new MechModel("mech");
        addModel(mech);

        // Create and add FemModel
        fem = new FemModel3d("fem");
        mech.add(fem);

        // Build hex beam using factory method
        FemFactory.createHexGrid {
            fem, length, width, width, /*nx=*/6, /*ny=*/3, /*nz=*/3);

        // Set FEM properties
        fem.setDensity(density);
    }
}
```

Figure 6.4: FemBeam model loaded into ArtiSynth.
This example can be found in artisynth.demos.tutorial.FemBeam. The build() method first creates a MechModel and FemModel3d. A FEM beam is created using a factory method on line 36. This beam is centered at the origin, so its length extends from \(-\text{length}/2\) to \text{length}/2. The density, damping and material properties are then assigned.

On lines 45–49, a fixed boundary condition is set to the left-hand side of the beam by setting the corresponding nodes to be non-dynamic. Due to numerical precision, a small \text{EPSILON} buffer is required to ensure all left-hand boundary nodes are identified (line 46).

Rendering properties are then assigned to the FEM model on line 52. These will be discussed further in Section 6.11.

### 6.3 FEM Geometry

Associated with each FEM model is a list of geometry with the heading meshes. This geometry can be used for either display purposes, or for interactions such as collisions (Section 7.3.2). A geometry itself has no physical properties; its motion is entirely governed by the FEM model that contains it.

All FEM geometries are of type FemMeshComp, which stores a reference to a mesh object (Section 2.5), as well as attachment information that links vertices of the mesh to points within the FEM. The attachments enforce the shape function interpolation in Equation (6.2) to hold at each mesh vertex, with constant shape function coefficients.

#### 6.3.1 Surface meshes

By default, every FemModel3d has an auto-generated geometry representing the “surface mesh”. The surface mesh consists of all un-shared element faces (i.e. the faces of individual elements that are exposed to the world), and its vertices correspond to the nodes that make up those faces. As the FEM nodes move, so do the mesh vertices due to the attachment framework.

The surface mesh can be obtained using one of the following functions in FemModel3d:

```java
FemMeshComp getSurfaceMeshComp (); // returns the FemMeshComp surface component
PolygonalMesh getSurfaceMesh (); // returns the underlying polygonal surface mesh
```

The first returns the surface complete with attachment information. The latter method directly returns the PolygonalMesh that is controlled by the FEM.

It is possible to manually set the surface mesh:

```java
setSurfaceMesh ( PolygonalMesh surface ); // manually set surface mesh
```
However, doing so is normally not necessary. It is always possible to add additional mesh geometries to a finite element model, and the visibility settings can be changed so that the default surface mesh is not rendered.

### 6.3.2 Embedding geometry within an FEM

Any geometry of type `MeshBase` can be added to a `FemModel3d`. To do so, first position the mesh so that its vertices are in the desired locations inside the FEM, then call one of the `FemModel3d` methods:

```
FemMeshComp addMesh (MeshBase mesh);  // creates and returns ←
FemMeshComp addMesh (String name, MeshBase mesh);
```

The latter is a convenience routine that also gives the newly embedded `FemMeshComp` a name.

### 6.3.3 Example: a beam with an embedded sphere

A complete model demonstrating embedding a mesh is given below.

```java
package artisynth.demos.tutorial;

import java.awt.Color;
import java.io.IOException;
import maspack.geometry.*;
import maspack.render.RenderProps;
import artisynth.core.mechmodels.Collidable.Collidability;
import artisynth.core.femmodels.*;
import artisynth.core.femmodels.FemModel.SurfaceRender;
import artisynth.core.materials.LinearMaterial;
import artisynth.core.mechmodels.MechModel;
import artisynth.core.workspace.RootModel;

public class FemEmbeddedSphere extends RootModel {
    // Internal components
    protected MechModel mech;
    protected FemModel3d fem;
    protected FemMeshComp sphere;

    @Override
    public void build(String[] args) throws IOException {
        super.build(args);
    }
}
```

![Figure 6.5: FemEmbeddedSphere model loaded into ArtiSynth.](image)

A complete model demonstrating embedding a mesh is given below.
This example can be found in artisynth.demos.tutorial.FemEmbeddedSphere. The model is very similar to FemBeam. A MechModel and FemModel3d are created and added. At line 41, a PolygonalMesh of a sphere is created using a factory method. The sphere is already centered inside the beam, so it does not need to be repositioned. At Line 42, the sphere is embedded inside model fem, creating a FemMeshComp with the name “sphere”. The full model is shown in Figure 6.5.

6.4 Connecting FEM models to other components

To couple FEM models to other dynamic components, the “attachment” mechanism described in Section 1.2 is used. This involves creating and adding to the model attachment components, which are instances of DynamicAttachment, as described in Section 3.6. Common point-based attachment classes are listed in Table 6.5.
### Table 6.5: Point-based attachments

<table>
<thead>
<tr>
<th>Attachment</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PointParticleAttachment</td>
<td>Attaches one “point” to one “particle”</td>
</tr>
<tr>
<td>PointFrameAttachment</td>
<td>Attaches one “point” to one “frame”</td>
</tr>
<tr>
<td>PointFem3dAttachment</td>
<td>Attaches one “point” to a linear combination of FEM nodes</td>
</tr>
</tbody>
</table>

FEM models are connected to other model components by attaching their nodes to various components. This can be done by creating an attachment object of the appropriate type, and then adding it to the MechModel using

```java
addAttachment (DynamicAttachment attach); // adds an attachment constraint
```

There are also convenience routines inside `MechModel` that will create the appropriate attachments automatically (see Section 3.6.1).

All attachments described in this section are based around FEM nodes. However, it is also possible to attach frame-based components (such as rigid bodies) directly to an FEM, as described in Section 6.6.

#### 6.4.1 Connecting nodes to rigid bodies or particles

Since `FemNode3d` is a subclass of `Particle`, the same methods described in Section 3.6.1 for attaching particles to other particles and frames are available. For example, we can attach an FEM node to a rigid body using a either a statement of the form

```java
mech.addAttachment (new PointFrameAttachment (body, node));
```

or the following equivalent statement which does the same thing:

```java
mech.attachPoint (node, body);
```

Both of these create a `PointFrameAttachment` between a rigid body (called `body`) and an FEM node (called `node`) and then adds it to the `MechModel mech`.

One can also attach the nodes of one FEM model to the nodes of another using statements like

```java
mech.addAttachment (new PointParticle (node1, node2));
```

or

```java
mech.attachPoint (node2, node1);
```

which attaches `node2` to `node1`.

#### 6.4.2 Example: connecting a beam to a block

The following model demonstrates attaching an FEM beam to a rigid block.

```java
package artisynth.demos.tutorial;
import java.io.IOException;
import maspack.matrix.RigidTransform3d;
import artisynth.core.femmodels.FemNode3d;
import artisynth.core.mechmodels.PointFrameAttachment;
import artisynth.core.mechmodels.RigidBody;

public class FemBeamWithBlock extends FemBeam {
    public void build (String[] args) throws IOException {
        // Build simple FemBeam
        super.build (args);
    }
}
```
This model extends the FemBeam example of Section 6.2.5. The build() method then creates and adds a RigidBody block (lines 18–20). On line 21, the block is repositioned to the side of the beam to prepare for the attachment. On lines 24–28, all right-most nodes of the beam are then set to be attached to the block using a PointFrameAttachment. In this case, the attachments are explicitly created. They could also have been attached using

```
mech.attachPoint (node, block); // attach node to rigid block
```

### 6.4.3 Connecting nodes directly to elements

Typically, nodes do not align in a way that makes it possible to connect them to other FEM models and/or points based on simple point-to-node attachments. Instead, we use a different mechanism that allows us to attach a point to an arbitrary location within an FEM model. This is done using an attachment component of type PointFem3dAttachment, which implements an attachment where the position \( p \) and velocity \( u \) of the attached point is determined by a weighted sum of the positions \( x_k \) and velocities \( u_k \) of selected fem nodes:

\[
p = \sum \alpha_k x_k
\]  

Any force \( f \) acting on the attached point is then propagated back to the nodes, according to the relation

\[
f_k = \alpha_k f'
\]

where \( f_k \) is the force acting on node \( k \) due to \( f \). This relation can be derived based on the conservation of energy. If \( p \) is embedded within a single element, then the \( x_k \) are simply the element nodes and the \( \alpha_k \) are corresponding shape function values; this is known as an element-based attachment. On the other hand, as described below, it is sometimes
desirable to form an attachment using a more general set of nodes that extends beyond a single element; this is known as a nodal-based attachment (Section 6.4.7).

An element-based attachment can be created using a code fragment of the form

```java
PointFem3dAttachment ax = new PointFem3dAttachment(pnt);
ax.setFromElement(pnt.getPosition(), elem);
mech.addAttachment(ax);
```

First, a `PointFem3dAttachment` is created for the point `pnt`. Next, `setFromElement()` is used to determine the nodal weights within the element `elem` for the specified position (which in this case is simply the point’s current position). To do this, it computes the “natural coordinates” coordinates of the position within the element. For this to be guaranteed to work, the position should be on or inside the element. If natural coordinates cannot be found, the method will return `false` and the nearest estimates coordinates will be used instead. However, it is sometimes possible to find natural coordinates outside a given element as long as the shape functions are well-defined. Finally, the attachment is added to the model.

More conveniently, the exact same functionality is provided by the `attachPoint()` method in `MechModel`:

```java
mech.attachPoint(pnt, elem);
```

This creates an attachment identical to that created by the previous code fragment.

Often, one does not want to have to determine the element to which a point should be attached. In that case, one can call

```java
PointFem3dAttachment ax = new PointFem3dAttachment(pnt);
ax.setFromFem(pnt.getPosition(), fem);
mech.addAttachment(ax);
```

or, equivalently,

```java
mech.attachPoint(pnt, fem);
```

This will find the nearest element to the node in question and use that to create the attachment. If the node is outside the FEM model, then it will be attached to the nearest point on the FEM’s surface.

### 6.4.4 Example: connecting two FEMs together

![Figure 6.7: FemBeamWithFemSphere model loaded into ArtiSynth.](image)

The following model demonstrates how to attach two FEM models together:

```java
package artisynth.demos.tutorial;
import java.io.IOException;
```
import maspack.matrix.RigidTransform3d;
import artisynth.core.femmodels. *
import artisynth.core.materials.LinearMaterial;
import maspack.util.PathFinder;

public class FemBeamWithFemSphere extends FemBeam {

    public void build (String[] args) throws IOException {

        // Build simple FemBeam
        super.build (args);

        // Create a FEM sphere
        FemModel3d femSphere = new FemModel3d("sphere");
        mech.addModel(femSphere);

        // Read from TetGen file
        TetGenReader.read(femSphere,
                         PathFinder.getSourceRelativePath(FemModel3d.class, "meshes/sphere2.1.node"),
                         PathFinder.getSourceRelativePath(FemModel3d.class, "meshes/sphere2.1.ele"));

        femSphere.scaleDistance(0.22);

        // FEM properties
        femSphere.setDensity(10);
        femSphere.setParticleDamping(2);
        femSphere.setMaterial(new LinearMaterial(4000, 0.33));

        // Reposition FEM to side of beam
        femSphere.transformGeometry( new RigidTransform3d(length/2+width/2, 0, 0) );

        // Attach sphere nodes that are inside beam
        for (FemNode3d node : femSphere.getNodes()) {
            // Find element containing node (if exists)
            FemElement3d elem = fem.findContainingElement(node.getPosition());

            // Add attachment if node is inside "fem"
            if (elem != null) {
                mech.attachPoint(node, elem);
            }
        }

        // Set render properties
        setRenderProps(femSphere);
    }
}

This example can be found in artisynth.demos.tutorial.FemBeamWithFemSphere. The model extends FemBeam, adding a finite element sphere and coupling them together. The sphere is created and added on lines 18–28. It is read from TetGen-generated files using the TetGenReader class. The model is then scaled to match the dimensions of the current model, and transformed to the right side of the beam. To create attachments, the code first checks for any nodes that belong to the sphere that fall inside the beam using the FemModel3d.findContainingElement(Point3d) method (line 36), which returns the containing element if the point is inside the model, or null if the point is outside. Internally, this spatial search uses a bounding volume hierarchy for efficiency (see BVTree and BVFeatureQuery). If the point is contained within the beam, then mech.attachPoint() is used to attach it to the nodes of the element (line 39).

6.4.5 Finding which nodes to attach

While it is straightforward to connect nodes to rigid bodies or other FEM nodes or elements, it is often necessary to determine which nodes to attach. This was evident in the example of Section 6.4.4, which attached nodes of an FEM sphere that were found to be inside an FEM beam.
As in that example, finding the nodes to attach can often be done using geometric queries. For example, we often select nodes based on how close they are to the other body we wish to attach to.

Various proliﬁxity queries are available for this task. To ﬁnd the distance of a point to a polygonal mesh, we can use the following PolygonalMesh methods,

\[
\begin{align*}
\text{double} & \quad \text{distanceToPoint (Point3d pnt)} \\
\text{int} & \quad \text{isInside (Point3d pnt)} \quad \text{// for closed meshes only}
\end{align*}
\]

where the latter method returns 1 if the point is inside and 0 otherwise. For checking the distance of an FEM node, \(pnt\) can be obtained from \(\text{node.getPosition()}\) (or possibly \(\text{node.getRestPosition()}\)). For example, to ﬁnd all nodes within a distance \(\text{tol}\) of the surface of a rigid body, we could use the code fragment:

```java
RigidBody body;
FemModel3d fem;
...
double tol = 0.001;
PolygonalMesh surface = body.getSurfaceMesh();
ArrayList<FemNode3d> nearNodes = new ArrayList<FemNode3d>();
for (FemNode3d n : fem.getNodes()) {
    if (surface.distanceToPoint (n.getPosition()) < tol) {
        nearNodes.add (n);
    }
}
```

If we want to check only nodes that lie on the FEM surface, then we can ﬁlter them using the FemModel3d method \(\text{isSurfaceNode()}\):

```java
for (FemNode3d n : fem.getNodes()) {
    if (fem.isSurfaceNode (n) &&
        surface.distanceToPoint (n.getPosition()) < tol) {
        nearNodes.add (n);
    }
}
```

Most of the mesh-based query methods work only for triangular meshes. The PolygonalMesh method \(\text{isTriangular()}\) can be used to determine if the mesh is triangular. If it is not, it can made triangular by calling \(\text{triangulate()}\), although in general this should be done during model construction before the mesh-based component has been added to the model.

For connecting an FEM model to another FEM model, FemModel3d provides a number of query methods:

```java
/* nearest element queries */
FemElement3dBase findNearestElement (Point3d near, Point3d pnt)
FemElement3dBase findNearestElement (Point3d near, Point3d pnt, ElementFilter filter)
FemElement3dBase findNearestSurfaceElement (Point3d near, Point3d pnt)
FemElement3d findNearestVolumetricElement (Point3d near, Point3d pnt)
ShellElement3d findNearestShellElement (Point3d near, Point3d pnt)
ShellElement3d findNearestShellElement (Point3d near, Point3d pnt, ElementFilter filter)
```

```java
/* nearest node queries */
```
All the above queries are based on the FEM model’s current nodal positions. The method
\texttt{findNearestElement(near, pnt, filter)} allows the application to specify a \texttt{FemModel.ElementFilter} to restrict the elements that are searched.

The argument \texttt{near} that appears in some of the queries is an optional argument which, if not \texttt{null}, returns the location of the corresponding nearest point on the element. The distance from \texttt{pnt} to the element can then be found using

\texttt{near.distance(pnt);}

If the resulting distance is 0, then the point is on or inside the element. Otherwise, the point is outside the element, and if no element filters were used in the query, outside the FEM model itself.

Typically, it is preferred attach a point to an element only if it lies on or inside an element. However, it is possible to attach points outside an element as long as the system is able to determine appropriate element “coordinates” for that point (which it may not be able to do if the point is far away). In addition, the motion behavior of an exterior attached point may sometimes appear counterintuitive.

The \texttt{FemModel3d} element and node queries can be used in a variety of ways. 
\texttt{findNearestNodes()} can be used to find all nodes within a certain distance of a point, as part of the process of making nodal-based attachments (Section 6.4.7).

\texttt{findNearestNode()} is used in the FemMuscleBeam example (Section 6.9.4) to determine if a desired muscle point is near enough to a node to use that node directly, or if a marker should be created.

As another example, suppose we wish to connect the surface nodes of an FEM model \texttt{femA} to the surface elements of another model \texttt{femB} if they lie within a prescribed distance \texttt{tol} of the surface of \texttt{femB}. Then we could use the following code:

```java
MechModel mech;
FemModel3d femA;
FemModel3d femB;
...
double tol = 0.001;
Point3d near = new Point3d();
for (FemNode3d n : femA.getNodes()) {
    if (femA.isSurfaceNode(n)) {
        FemElement3dBase elem = femB.findNearestSurfaceElement(near, n.getPosition());
        if (elem != null && near.distance(n.getPosition()) <= tol) {
            // attach if within distance
            mech.attachPoint(n, elem);
        }
    }
}
```

### 6.4.6 Example: two bodies connected by an FEM “spring”

The \texttt{LumbarFrameSpring} example in Section 3.5.4 uses a frame spring to connect two simplified lumbar vertebrae. However, it is also possible to use an FEM model in place of a frame spring, possibly providing a more realistic model of the intervertebral disc. A simple model which does this is defined in

\texttt{artisynth.demos.tutorial.LumbarFEMDisk}
The initial source code is similar to that for LumbarFrameSpring, but differs in the section where the FEM disk replaces the FrameSpring:

```java
// create a torus shaped FEM model for the disk
FemModel3d fem = new FemModel3d();
fem.setDensity (1500);
fem.setMaterial (new LinearMaterial (20000, 0.4));
FemFactory.createHexTorus (fem, 0.011, 0.003, 0.008, 16, 30, 2);
mech.addModel (fem);

// position it between the disks
double DTOR = Math.PI/180.0;
fem.transformGeometry (new RigidTransform3d (-0.012, 0.0, 0.040, 0, -DTOR*25, DTOR*90));

// find and attach nearest nodes to either the top or bottom mesh
double tol = 0.001;
for (FemNode3d n : fem.getNodes()) {
    // top vertebra
    double d = lumbar1.getSurfaceMesh().distanceToPoint (n.getPosition());
    if (d < tol) {
        mech.attachPoint (n, lumbar1);
    }
    // bottom vertebra
    d = lumbar2.getSurfaceMesh().distanceToPoint (n.getPosition());
    if (d < tol) {
        mech.attachPoint (n, lumbar2);
    }
}

// set render properties ...
fem.setSurfaceRendering (SurfaceRender.Shaded);
RenderProps.setFaceColor (fem, new Color (153/255f, 153/255f, 1f));
RenderProps.setFaceColor (mech, new Color (238, 232, 170)); // bone color
```

The simplified FEM model representing the “disk” is created at lines 57-61, using a torus-shaped model created by FemFactory. It is then repositioning using transformGeometry() (Section 4.6) to place it between the vertebrae (line 64-66). After the FEM model is positioned, we find which nodes are within a distance tol of each vertebral surface and attach them to the appropriate body (lines 69-81).

To run this example in ArtiSynth, select All demos > tutorial > LumbarFEMDisk from the Models menu. The model
should load and initially appear as in Figure 6.8. The behavior is best seen by running the model and using the pull controller to exert forces on the upper vertebra.

### 6.4.7 Nodal-based attachments

The example of Section 6.4.4 uses element-based attachments to connect the nodes of one FEM to elements of another. As mentioned above, element-based attachments assume that the attached point is associated with a specific FEM model element. While this often gives good results, there are situations where it may be desirable to distribute the connection more broadly among a larger set of nodes.

In particular, this is sometimes the case when connecting FEM models to point-to-point springs. The end-point of such a spring may end up exerting a large force on the FEM, and then if the number of nodes to which the end-point is attached are too small, the resulting forces on these nodes (Equation 6.5) may end up being too large. In other words, it may be desirable to distribute the spring’s force more evenly throughout the FEM model.

To handle such situations, it is possible to create a nodal-based attachment in which the nodes and weights are explicitly specified. This involves explicitly creating a `PointFem3dAttachment` for the point or particle to be attached and the specifying the nodes and weights directly:

```java
PointFem3dAttachment ax = new PointFem3dAttachment (part);
ax.setFromNodes (nodes, weights);
mech.addAttachment (ax);
```

where `nodes` and `weights` are arrays of `FemNode` and `double`, respectively. It is up to the application to determine these.

`PointFem3dAttachment` provides several methods for explicitly specifying nodes and weights. The signatures for these include:

```java
void setFromNodes (FemNode[] nodes, double[] weights)
void setFromNodes (Collection<FemNode> nodes, VectorNd weights)
boolean setFromNodes (Point3d pos, FemNode[] nodes)
boolean setFromNodes (Point3d pos, Collection<FemNode> nodes)
```

The last two methods determine the weights automatically, using an inverse-distance-based calculation in which each weight $\alpha_k$ is initially computed as

$$
\alpha_k = \frac{d_{\text{max}}}{d_k + d_{\text{max}}}
$$

where $d_k$ is the distance from node $k$ to `pos` and $d_{\text{max}}$ is the maximum distance. The weights are then adjusted to ensure that they sum to one and that the weighted sum of the nodes equals `pos`. In some cases, the specified nodes may not provide enough support for the last condition to be met, in which case the methods return `false`.

### 6.4.8 Example: element vs. nodal-based attachments

The model demonstrating the difference between element and nodal-based attachments is defined in

```java
artisynth.demos.tutorial.PointFemAttachment
```

It creates two FEM models, each with a single point-to-point spring attached to a particle at their center. The model at the top (`fem1` in the code below) is connected to the particle using an element-based attachment, while the lower model (`fem2` in the code) is connected using a nodal-based attachment with a larger number of nodes. Figure 6.9 shows the result after the model is run until stable. The element-based attachment results in significantly higher deformation in the immediate vicinity around the attachment, while for the nodal-based attachment, the deformation is distributed much more evenly through the model.

The build method and some of the auxiliary code for this model is shown below. Code for the other auxiliary methods, including `addFem()`, `addParticle()`, `addSpring()`, and `setAttachedNodesWhite()`, can be found in the actual source file.
Figure 6.9: PointFemAttachment loaded into ArtiSynth and run until stable. The top and bottom models are connected to their springs using element and nodal-based attachments, respectively. The nodes associated with each attachment are rendered as white spheres.

```java
// Filter to select only elements for which the nodes are entirely on the positive side of the x-z plane.
private class MyFilter extends ElementFilter {
    public boolean elementIsValid(FemElement e) {
        for (FemNode n : e.getNodes()) {
            if (n.getPosition().y < 0) {
                return false;
            }
        }
        return true;
    }
}

// Collect and return all the nodes of an FEM model associated with a set of elements specified by an array of element numbers
private HashSet<FemNode3d> collectNodes(FemModel3d fem, int[] elemNums) {
    HashSet<FemNode3d> nodes = new HashSet<FemNode3d>();
    for (int i=0; i<elemNums.length; i++) {
        FemElement3d e = fem.getElements().getByNumber(elemNums[i]);
        for (FemNode3d n : e.getNodes()) {
            nodes.add(n);
        }
    }
    return nodes;
}

public void build(String[] args) {
    MechModel mech = new MechModel("mech");
    addModel(mech);
    mech.setGravity(0, 0, 0); // turn off gravity
    // create and add two FEM beam models centered at the specified locations
    FemModel13d fem1 = addFem(mech, 0.0, 0.0, 0.25);
    FemModel13d fem2 = addFem(mech, 0.0, 0.0, -0.25);
    // reconstruct the FEM surface meshes so that they show only elements on the positive side of the x-y plane. Also, set surface rendering to show strain values.
```
The `build()` method begins by creating a `MechModel` and then adding to it two FEM beams (created using the auxiliary method `addFem()`). Rendering of each FEM model’s surface is then set up to show strain values (`setSurfaceRendering()`, lines 41 and 43). The surface meshes themselves are also redefined to exclude the frontmost elements, allowing the strain values to be displayed closer model centers. This redefinition is done using calls to `createSurfaceMesh()` (lines 40, 41) with a custom `ElementFilter` defined at lines 3-12.

Next, the end-point particles for the axial springs are created (using the auxiliary method `addParticle()`, lines 46-49), and particle `m1` is attached to `fem1` using `mech.attachPoint()` (line 52), which creates an element-based attachment at the point’s current location. Point `m2` is then attached to `fem2` using a nodal-based attachment. The nodes for these are collected as the union of all nodes for a specified set of elements (lines 58-59, and the method `collectNodes()` defined at lines 16-25). These are then used to create a nodal-based attachment (lines 61-63), where the weights are determined automatically using the method associated with equation (6.6).

Finally, the springs are created (auxiliary method `addSpring()`, lines 66-67), the nodes associated for each attachment are set to render as white spheres (`setAttachedNodesWhites()`, lines 70-71), and the particles are set to render as green spheres.

To run this example in ArtiSynth, select `All demos > tutorial > PointFemAttachment` from the `Models` menu. Running the model will cause it to settle into the state shown in Figure 6.9. Selecting and dragging one of the spring anchor points at the right will cause the spring tension to vary and further illustrate the difference between the element and nodal-based attachments.

6.5 FEM markers

Just as there are `FrameMarkers` to act as anchor points on a frame or rigid body (Section 3.2.1), there are also `FemMarkers` that can mark a point inside a finite element. They are frequently used to provide anchor points for
attaching springs and forces to a point inside an element, but can also be used for graphical purposes.

FEM markers are implemented by the class FemMarker, which is a subclass of Point. They are essentially massless points that contain their own attachment component, so when creating and adding a marker there is no need to create a separate attachment component.

Within the component hierarchy, FEM markers are typically stored in the markers list of their associated FEM model. They can be created and added using a code fragment of the form

```java
FemMarker mkr = new FemMarker(1, 0, 0);
mkr.setFromFem(fem); // attach to the nearest fem element
fem.addMarker(mkr); // add to fem
```

This creates a marker at the location (1, 0, 0) (in world coordinates), calls setFromFem() to attach it to the nearest element in the FEM model (which is either the containing element or the nearest element on the model’s surface), and then adds it to the markers list.

If the marker’s attachment has not already been set when addMarker() is called, then addMarker() will call setFromFem() automatically. Therefore the above code fragment is equivalent to the following:

```java
FemMarker mkr = new FemMarker(1, 0, 0);
fem.addMarker(mkr);
```

Alternatively, one may want to explicitly specify the nodes associated with the attachment, as described in Section 6.4.7:

```java
FemMarker mkr = new FemMarker(1, 0, 0);
mkr.setFromNodes(nodes, weights);
fem.addMarker(mkr);
```

There are a variety of methods available to set the attachment, mirroring those available in the underlying base class PointFem3dAttachment:

```java
void setFromFem (FemModel3d fem)
boolean setFromElement (FemElement3d elem)
void setFromNodes (FemNode[] nodes, double[] weights)
void setFromNodes (Collection<FemNode> nodes, VectorNd weights)
boolean setFromNodes (FemNode[] nodes)
boolean setFromNodes (Collection<FemNode> nodes)
```

The last two methods compute nodal weights automatically, as described in Section 6.4.7, based on the marker’s currently assigned position. If the supplied nodes do not provide sufficient support, then the methods return false.

Another set of convenience methods are supplied by FemModel3d, which combine these with the addMarker() call:

```java
void addMarker (FemMarker mkr, FemElement3d elem)
void addMarker (FemMarker mkr, FemNode[] nodes, double[] weights)
void addMarker (FemMarker mkr, Collection<FemNode> nodes, VectorNd weights)
boolean addMarker (FemMarker mkr, FemNode[] nodes)
boolean addMarker (FemMarker mkr, Collection<FemNode> nodes)
```

For example, one can do

```java
FemMarker mkr = new FemMarker(1, 0, 0);
fem.addMarker(mkr, nodes, weights);
```

Markers are often used to track movement within an FEM model. For that, one can examine their positions and velocities, as with any other particles, using the methods

```java
Point3d getPosition(); // returns the current position
Vector3d getVelocity(); // returns the current velocity
```

The return values from these methods should not be modified. Alternatively, when a 3D force $f$ is applied to the marker, it is distributed to the attached nodes according to the nodal weights, as described in Equation (6.5).
6.5.1 Example: attaching an FEM beam to a muscle

A complete application model that employs a fem marker as an anchor for a spring is given below.

```java
package artisynth.demos.tutorial;

import java.awt.Color;
import java.io.IOException;

import maspack.render.RenderProps;
import maspack.render.Renderer;
import artisynth.core.femmodels.FemMarker;
import artisynth.core.femmodels.FemModel3d;
import artisynth.core.materials.SimpleAxialMuscle;
import artisynth.core.mechmodels.Muscle;
import artisynth.core.mechmodels.Particle;
import artisynth.core.mechmodels.Point;

public class FemBeamWithMuscle extends FemBeam {

  // Creates a point-to-point muscle
  protected Muscle createMuscle() {
    Muscle mus = new Muscle(/* name = */null, /* restLength = */0);
    new SimpleAxialMuscle(/* stiffness = */20, /* damping = */10,/* maxf = */10);
    RenderProps.setLineStyle(mus, Renderer.LineStyle.SPINDLE);
    RenderProps.setLineColor(mus, Color.RED);
    RenderProps.setLineRadius(mus, 0.03);
    return mus;
  }

  // Creates a FEM Marker
  protected FemMarker createMarker(FemModel3d fem, double x, double y, double z) {
    FemMarker mkr = new FemMarker( /* name = */null, x, y, z);
    RenderProps.setSphericalPoints(mkr, 0.02, Color.BLUE);
    fem.addMarker(mkr);
    return mkr;
  }

  public void build(String[] args) throws IOException {
```

Figure 6.10: FemBeamWithMuscle model loaded into ArtiSynth.
// Create simple FEM beam
super.build (args);

// Add a particle fixed in space
Particle p1 = new Particle /*mass=*/0, -length/2, 0, 2*width);
mech.addParticle (p1);
p1.setDynamic (false);
RenderProps.setSphericalPoints (p1, 0.02, Color.BLUE);

// Add a marker at the end of the model
FemMarker mkr = createMarker (fem, length/2-0.1, 0, width/2);

// Create a muscle between the point an marker
Muscle muscle = createMuscle();
muscle.setPoints (p1, mkr);
mech.addAxialSpring (muscle);
}

This example can be found in artisynth.demos.tutorial.FemBeamWithMuscle. This model extends the FemBeam example, adding a FemMarker for the spring to attach to. The method createMarker(...) on lines 29–35 is used to create and add a marker to the FEM. Since the element is initially set to null, when it is added to the FEM, the model searches for the containing or nearest element. The loaded model is shown in Figure 6.10.

6.6 Frame attachments

It is also possible to attach frame components, including rigid bodies, directly to FEM models, using the attachment component FrameFem3dAttachment. Analogously to PointFem3dAttachment, the attachment is implemented by connecting the frame to a set of FEM nodes, and attachments can be either element-based or nodal-based. The frame’s origin is computed in the same way as for point attachments, using a weighted sum of node positions (Equation 6.4), while the orientation is computed using a polar decomposition on a deformation gradient determined from either element shape functions (for element-based attachments) or a Procrustes type analysis using nodal rest positions (for nodal-based attachments).

An element-based attachment can be created using either a code fragment of the form

```java
FrameFem3dAttachment ax = new FrameFem3dAttachment (frame);
ax.setFromElement (frame.getPose (), elem);
mech.addAttachment (ax);
```
or, equivalently, the attachFrame() method

```java
mech.attachFrame (frame, elem);
```

This attaches the frame frame to the nodes of the FEM element elem. As with PointFem3dAttachment, if the frame’s origin is not inside the element, it may not be possible to accurately compute the internal nodal weights, in which case setFromElement() will return false.

In order to have the appropriate element located automatically, one can instead use

```java
FrameFem3dAttachment ax = new FrameFem3dAttachment (frame);
ax.setFromFem (frame.getPose (), fem);
mech.addAttachment (ax);
```
or, equivalently,

```java
mech.attachFrame (frame, fem);
```

As with point-to-FEM attachments, it may be desirable to create a nodal-based attachment in which the nodes and weights are not tied to a specific element. The reasons for this are generally the same as with nodal-based point
attachments (Section 6.4.7): the need to distribute the forces and moments acting on the frame across a broader set of element nodes. Also, element-based frame attachments use element shape functions to determine the frame’s orientation, which may produce slightly asymmetric results if the frame’s origin is located particularly close to a specific node.

FrameFem3dAttachment provides several methods for explicitly specifying nodes and weights. The signatures for these include:

```java
void setFromNodes (RigidTransform3d TFW, FemNode[] nodes, double[] weights)
void setFromNodes (RigidTransform3d TFW, Collection<FemNode> nodes, VectorNd weights)
boolean setFromNodes (RigidTransform3d TFW, FemNode[] nodes)
boolean setFromNodes (RigidTransform3d TFW, Collection<FemNode> nodes)
```

Unlike their counterparts in PointFem3dAttachment, the first two methods also require the current desired pose of the frame TFW (in world coordinates). This is because while nodes and weights will unambiguously specify the frame’s origin, they do not specify the desired orientation.

### 6.6.1 Example: attaching frames to an FEM beam

A model illustrating how to connect frames to an FEM model is defined in `artisynth.demos.tutorial.FrameFemAttachment` It creates an FEM beam, along with a rigid body block and a massless coordinate frame, that are then attached to the beam using nodal and element-based attachments. The build method is shown below:

```java
public void build (String[] args) {
  MechModel mech = new MechModel (*mech*);
  addModel (mech);

  // create and add FEM beam
  FemModel3d fem = FemFactory.createHexGrid (null, 1.0, 0.2, 0.2, 6, 3, 3);
  fem.setMaterial (new LinearMaterial (500000, 0.33));
  RenderProps.setLineColor (fem, Color.BLUE);
  RenderProps.setLineWidth (mech, 2);
  mech.addModel (fem);

  // fix leftmost nodes of the FEM
  for (FemNode3d n : fem.getNodes()) {
```
if (n.getPosition().x-(-0.5)) < 1e-8) {
    n.setDynamic(false);
}

// create and add rigid body box
RigidBody box = RigidBody.createBox("box", 0.25, 0.1, 0.1, /*density=*/1000);
mech.add(box);

// create a basic frame and set its pose and axis length
Frame frame = new Frame();
frame.setPose(new RigidTransform3d(0.4, 0, 0, Math.PI/4, 0));
frame.setAxisLength(0.3);
mech.addFrame(frame);
mech.attachFrame(frame, fem); // attach using element-based attachment

// attach the box to the FEM, using all the nodes of elements 31 and 32
HashSet<FemNode3d> nodes = collectNodes(fem, new int[] {22, 31});
FrameFem3dAttachment attachment = new FrameFem3dAttachment(box);
attachment.setFromNodes(box.getPose(), nodes);
mech.addAttachment(attachment);

// render the attachment nodes for the box as spheres
for (FemNode n : attachment.getNodes()) {
    RenderProps.setSphericalPoints(n, 0.007, Color.GREEN);
}

Lines 3-22 create a MechModel and populate it with an FEM beam and a rigid body box. Next, a basic Frame is created, with a specified pose and an axis length of 0.3 (to allow it to be seen), and added to the MechModel (lines 25-28). It is then attached to the FEM beam using an element-based attachment (line 30). Meanwhile, the box is attached to using a nodal-based attachment, created from all the nodes associated with elements 22 and 31 (lines 33-36). Finally, all attachment nodes are set to be rendered as green spheres (lines 39-41).

To run this example in ArtiSynth, select All demos > tutorial > FrameFemAttachment from the Models menu. Running the model will cause it to settle into the state shown in Figure 6.11. Forces can interactively be applied to the attached block and frame using the pull tool, causing the FEM model to deform (see the section “Pull Manipulation” in the ArtiSynth User Interface Guide).

### 6.6.2 Adding joints to FEM models

The ability to connect frames to FEM models, as described in Section 6.6, makes it possible to interconnect different FEM models directly using joints, as described in Section 3.3. This is done internally by using FrameFem3dAttachments to connect frames C and D of the joint (Figure 3.8) to their respective FEM models.

As indicated in Section 3.3.3, most joints have a constructor of the form

```java
JointType (bodyA, bodyB, TDW);
```

that creates a joint connecting bodyA to bodyB, with the initial pose of the D frame given (in world coordinates) by TDW. The same body and transform settings can be made on an existing joint using the method setBodies(bodyA, bodyB, TDW). For these constructors and methods, it is possible to specify FEM models for bodyA and/or bodyB. Internally, the joint then creates a FrameFem3dAttachment to connect frame C and/or D of the joint (See Figure 3.8) to the corresponding FEM model.

However, unlike joints involving rigid bodies or frames, there are no associated $T_{CA}$ or $T_{DB}$ transforms (since there is no fixed frame within an FEM to define such transforms). Methods or constructors which utilize $T_{CA}$ or $T_{DB}$ can therefore not be used with FEM models.
6.6.3 Example: two FEM beams connected by a joint

A model connecting two FEM beams by a joint is defined in

artisynth.demos.tutorial.JointedFemBeams

It creates two FEM beams and connects them via a special slotted-revolute joint. The build method is shown below:

```java
public void build (String[] args) {
    MechModel mech = new MechModel("mechMod");
    addModel(mech);

    double stiffness = 5000;
    // create first fem beam and fix the leftmost nodes
    FemModel3d fem1 = addFem(mech, 2.4, 0.6, 0.4, stiffness);
    for (FemNode3d n : fem1.getNodes()) {
        if (n.getPosition().x <= -1.2) {
            n.setDynamic(false);
        }
    }
    // create the second fem beam and shift it 1.5 to the right
    FemModel3d fem2 = addFem(mech, 2.4, 0.4, 0.4, 0.1*stiffness);
    fem2.transformGeometry(new RigidTransform3d(1.5, 0, 0));

    // create a slotted revolute joint that connects the two fem beams
    RigidTransform3d TDW = new RigidTransform3d(0.5, 0, 0, 0, Math.PI/2);
    SlottedRevoluteJoint joint = new SlottedRevoluteJoint(fem2, fem1, TDW);
    mech.addBodyConnector(joint);

    // set ranges and rendering properties for the joint
    joint.setShaftLength(0.8);
    joint.setMinX(-0.5);
    joint.setMaxX(0.5);
    joint.setSlotDepth(0.63);
    joint.setSlotWidth(0.08);
    RenderProps.setFaceColor(joint, myJointColor);
}
```

Lines 3-16 create a MechModel and populates it with two FEM beams, fem1 and fem2, using an auxiliary method addFem() defined in the model source file. The leftmost nodes of fem1 are set fixed. A SlottedRevoluteJoint is then
created to interconnect `fem1` and `fem2` at a location specified by `TDW` (lines 19-21). Lines 24-29 set some parameters for the joint, along with various render properties.

To run this example in ArtiSynth, select **All demos > tutorial > JointedFemBeams** from the **Models** menu. Running the model will cause it drop and flex under gravity, as shown in 6.12. Forces can interactively be applied to the beams using the pull tool (see the section “Pull Manipulation” in the **ArtiSynth User Interface Guide**).

### 6.7 Incompressibility

FEM incompressibility within ArtiSynth is enforced by trying to ensure that the volume of an FEM remains locally constant. This, in turn, is accomplished by constraining nodal velocities so that the local volume change, or divergence, is zero (or close to zero). There are generally two ways to do this:

- **Hard incompressibility**, which sets up explicit constraints on the nodal velocities;
- **Soft incompressibility**, which uses a restoring pressure based on a potential field to try to keep the volume constant.

Both of these methods operate independently, and both can be used either separately or together. Generally speaking, hard incompressibility will result in incompressibility being more rigorously enforced, but at the cost of increased computation time and (sometimes) less stability. Soft incompressibility allows the application to control the restoring force used to enforce incompressibility, usually by adjusting the value of the *bulk modulus* material property. As the bulk modulus is increased, soft incompressibility starts to act more like ‘hard’ incompressibility, with an infinite bulk modulus corresponding to perfect incompressibility. However, very large bulk modulus values will generally produce stability problems.

Incompressibility is not currently implemented for shell elements. Applying hard incompressibility to a shell element will have no effect on its behavior. If soft incompressibility is applied, by supplying the element with an incompressible material, then only the deviatoric component of that material will have any effect; the dilational component will generate no stress.

### 6.7.1 Volume regions and locking

Both hard and soft incompressibility can be applied to different regions of local volume. From larger to smaller, these regions are:

- **Nodal** - the local volume surrounding each node;
- **Element** - the volume of each element;
- **Full** - the volume at each integration point.

Element-based incompressibility is the standard method generally seen in the literature. However, it tends not to work well for tetrahedral meshes, because constraining the volume of each tet in a tetrahedral mesh tends to over constrain the system. This is because the number of tets in a large tetrahedral mesh is often $O(5n)$, where $n$ is the number of nodes, and so putting a volume constraint on each element may result in $O(5n)$ constraints, which exceeds the $3n$ degrees of freedom (DOF) in the FEM. This overconstraining results in an artificially increased stiffness known as *locking*. Because of locking, for tetrahedrally based meshes it may be better to use nodal-based incompressibility, which creates a single volume constraint around each node, resulting in only $n$ constraints, leaving $2n$ DOF to handle the remaining deformation. However, nodal-based incompressibility is computationally more costly than element-based and may not be as stable.

Generally, the best solution for incompressible problems is to use element-based incompressibility with a mesh consisting of hexahedra, or primarily hexahedra and a mix of other elements (the latter commonly being known as a *hex dominant* mesh). For hex-based meshes, the number of elements is roughly equal to the number of nodes, and so adding a volume constraint for each element imposes $n$ constraints on the model, which (like nodal incompressibility) leaves $2n$ DOF to handle the remaining deformation.

Full incompressibility tries to control the volume at each integration point within each element, which almost always results in a large number of volumetric constraints and hence locking. It is therefore not commonly used and is provided mostly for debugging and diagnostic purposes.
### 6.7.2 Hard incompressibility

Hard incompressibility is controlled by the incompressible property of the FEM, which can be set to one of the following values of the enumerated type `FemModel.IncompMethod`:

- **OFF** No hard incompressibility enforced.
- **ELEMENT** Element-based hard incompressibility enforced (Section 6.7.1).
- **NODAL** Nodal-based hard incompressibility enforced (Section 6.7.1).
- **AUTO** Selects either **ELEMENT** or **NODAL**, with the former selected if the number of elements is less than or equal to the number of nodes.
- **ON** Same as **AUTO**.

Hard incompressibility uses explicit constraints on the nodal velocities to enforce the incompressibility, which increases computational cost. Also, if the number of constraints is too large, *perturbed pivot* errors may be encountered by the solver. However, hard incompressibility can in principle handle situations where complete incompressibility is required. It is equivalent to the mixed u-P formulation used in commercial FEM codes (such as ANSYS), and the Lagrange multipliers computed for the constraints are pressure impulses.

Hard incompressibility can be applied in addition to soft incompressibility, in which case it will provide additional incompressibility enforcement on top of that provided by the latter. It can also be applied to linear materials, which are not themselves able to emulate true incompressible behavior (Section 6.7.4).

### 6.7.3 Soft incompressibility

Soft incompressibility enforces incompressibility using a restoring pressure that is controlled by a volume-based energy potential. It is only available for FEM materials that are subclasses of `IncompressibleMaterial`. The energy potential $U(J)$ is a function of the determinant $J$ of the deformation gradient, and is scaled by the material’s *bulk modulus* $\kappa$. The restoring pressure $p$ is given by

$$p = \frac{\partial U}{\partial J}.$$  \hspace{1cm} (6.7)

Different potentials can be selected by setting the `bulkPotential` property of the incompressible material, whose value is an instance of `IncompressibleMaterial.BulkPotential`. Currently there are two different potentials:

- **QUADRATIC** The potential and associated pressure are given by

  $$U(J) = \frac{1}{2} \kappa (J - 1)^2, \quad p = \kappa (J - 1).$$  \hspace{1cm} (6.8)

- **LOGARITHMIC** The potential and associated pressure are given by

  $$U(J) = \frac{1}{2} \kappa (\ln J)^2, \quad p = \kappa \frac{\ln J}{J}.$$  \hspace{1cm} (6.9)

The default potential is **QUADRATIC**, which may provide slightly improved stability characteristics. However, we have not noticed significant differences between the two potentials in practice.

How soft incompressibility is applied within an FEM model is controlled by the FEM’s `softIncompMethod` property, which can be set to one of the following values of the enumerated type `FemModel.IncompMethod`:

- **ELEMENT** Element-based soft incompressibility enforced (Section 6.7.1).
- **NODAL** Nodal-based soft incompressibility enforced (Section 6.7.1).
- **AUTO** Selects either **ELEMENT** or **NODAL**, with the former selected if the number of elements is less than or equal to the number of nodes.
- **FULL** Incompressibility enforced at each integration point (Section 6.7.1).
6.7.4 Incompressibility and linear materials

Within a linear material, incompressibility is controlled by Poisson’s ratio \( \nu \), which for isotropic materials can assume a value in the range \([-1, 0.5]\). This specifies the amount of transverse contraction (or expansion) exhibited by the material as it compressed or extended along a particular direction. A value of 0 allows the material to be compressed or extended without any transverse contraction or expansion, while a value of 0.5 in theory indicates a perfectly incompressible material. However, setting \( \nu = 0.5 \) in practice causes a division by zero, so only values close to 0.5 (such as 0.49) can be used.

Moreover, the incompressibility only applies to small displacements, so that even with \( \nu = 0.49 \) it is still possible to squash a linear FEM completely flat if enough force is applied. If true incompressible behavior is desired with a linear material, then one must also use hard incompressibility (Section 6.7.2).

6.7.5 Using incompressibility in practice

As mentioned above, when modeling incompressible models, we have found that the best practice is to use, if possible, either a hex or hex-dominant mesh, along with element-based incompressibility.

Hard incompressibility allows the handling of full incompressibility but at the expense of greater computational cost and often less stability. When modeling biomechanical materials, it is often permissible to use only soft incompressibility, partly since biomechanical materials are rarely completely incompressible. When implementing soft incompressibility, it is common practice to set the bulk modulus to something like 100 times the other (deviatoric) stiffnesses of the material.

We have found stability behavior to be complex, and while hard incompressibility often results in less stable behavior, this is not always the case: in some situations the stronger enforcement afforded by hard incompressibility actually improves stability.

6.8 Varying and augmenting material behaviors

The default material used by all elements of an FEM model is supplied by the model’s material property. However, it is often the case that one wishes to specify different material behaviors for different sets of elements within an FEM model. This may be particularly true when combining volumetric and shell elements.

There are several ways to vary material behavior within a model. These include:

- **Setting an explicit material for specific elements**, using their material property. An element’s material is null by default, but if set to a material, it will override the default material supplied by the FEM model. While this method is quite straightforward, it does have one disadvantage: because material settings are copied by each element, subsequent interactive or online changes to the material require resetting the material in all the affected elements.

- **Binding one or more material parameters to a field**. Sometimes certain material parameters, such as stiffness quantities or direction information, may need to vary across the FEM domain. While sometimes this can be handled by setting material properties for specific elements, it may be more convenient to bind the varying properties to a field, which can specify varying values over a domain composed of either a regular grid or an FEM mesh. Only one material needs to be used, and any properties which are not bound can be adjusted interactively or online. Fields and their bindings are described in detail in the Section 6.10.

- **Adding augmenting material behaviors using MaterialBundles**. A material bundle may be specified either for all elements, or for a subset of them, and each provides one material (via its own material property) whose behavior is added to that of the indicated elements. This also provides an easy way to combine the behaviors of two or more materials in the same element. One also has the option of setting the material property of the certain elements to NullMaterial, so that only the augmenting material(s) are applied.

- **Adding muscle behaviors using MuscleBundles**. This is analogous to using MaterialBundles, except that MuscleBundles are restricted to using an instance of a MuscleMaterial, and include support for handling the excitation value, as well as the activation directions (which usually vary across the FEM domain). MuscleBundles are only present in the FemMuscleModels subclass of FemModel3d, and are described in detail in Section 6.9.
The remainder of this section will discuss `MaterialBundles`.

Adding a `MaterialBundle` to an FEM model is illustrated by the following code fragment:

```java
FemMaterial extraMat; // material to be added
FemModel3d fem; // FEM model
...

MaterialBundle bun = new MaterialBundle("mybundle", extraMat);
// add volumetric elements to the bundle
for (FemElement3d e : fem.getElements()) {
    if (/* e should be added to the bundle */) {
        bun.addElement(e);
    }
}
fem.addMaterialBundle(bun);
```

Once added, the stress computed by the bundle’s material will be *added* to the stress computed by any other materials which are active for the bundle’s elements.

When deciding what elements to add to a bundle, one is free to choose any means necessary. The example above inspects all the volumetric elements in the FEM. To instead inspect all the shell elements, or all volumetric and shell elements, one could use the code fragments such as the following:

```java
// add shell elements to the bundle
for (ShellElement3d e : fem.getShellElements()) {
    if (/* e should be added to the bundle */) {
        bun.addElement(e);
    }
}

// add volumetric or shell elements to the bundle
for (FemElement3dBase e : fem.getAllElements()) {
    if (/* e should be added to the bundle */) {
        bun.addElement(e);
    }
}
```

Of course, if the elements are known through other means, then they can be added directly.

The element composition of a bundle can be controlled by the methods

```java
void addElement (FemElement3dBase e) // adds an element
boolean removeElement (FemElement3dBase e) // removes an element
void clearElements() // removes all elements
int numElements() // gets the number of elements
FemElement3dBase getElement (int idx) // gets the idx-th element
```

It is also possible to create a `MaterialBundle` whose material is added to *all* the FEM elements. This can done either by using one of the following constructors

```java
MaterialBundle (String name, boolean useAllElements)
MaterialBundle (String name, FemMaterial mat, boolean useAllElements)
```

with `useAllElements` set to `true`, or by calling the method

```java
void setUseAllElements (boolean enable)
```

When a bundle is set to use all the FEM elements, it clears its own element list, and one is not permitted to add elements using `addElement(elem)`.

After a bundle has been created, it is possible to get or set its material property using the methods

```java
FemMaterial getMaterial() // get the material
FemMaterial setMaterial (FemMaterial mat) // set the material
```
Again, because materials are copied internally, any modification to a material after it has been used as an input to `setMaterial()` will not be noticed by the bundle. Instead, one should modify the material before calling `setMaterial()`, or modify the copied material which can be obtained by calling `getMaterial()` or by storing the value returned by `setMaterial()`.

Finally, a `MuscleBundle` is a renderable component. Setting its `elementWidgetSize` property to a value greater than zero will cause the rendering of all its elements, using a solid widget representation of each at a scale controlled by `elementWidgetSize`: 0.5 for half size, 1.0 for full size, etc. The color of the widgets is controlled by the `faceColor` property of the bundle’s `renderProps`. Being able to render the elements makes it easy to select the bundle and visualize which elements it contains.

### 6.8.1 Example: FEM sheet with a stiff spine

A simple model demonstrating the use of material bundles is defined in

`artisynth.demos.tutorial.MaterialBundleDemo`

It consists of a simple thin hexahedral sheet in which a material bundle is used to stiffen elements close to the x-axis, creating a stiff “spine”. While the same effect could be achieved by simply setting a different material property for the “spine” elements, it does provide a good example of muscle bundle usage. The model’s `build` method is given below:

```java
public void build (String[] args) {
    MechModel mech = new MechModel("mech");
    addModel(mech);
    // create a fem model consisting of a thin sheet of hexes
    FemModel3d fem = FemFactory.createHexGrid (null, 1.0, 1.0, 0.1, 10, 10, 1);
    fem.setDensity (1000);
    fem.setMaterial (new LinearMaterial (10000, 0.45));
    mech.add (fem);
    // fix the left-most nodes:
    double EPS = 1e-8;
    for (FemNode3d n : fem.getNodes()) {
        if (n.getPosition().x <= -0.5+EPS) {
            n.setDynamic (false);
        }
    }
    // create a "spine" of stiffer elements using a MaterialBundle with a
    // stiffer material
    MaterialBundle bun =
```
new MaterialBundle ("spine", new NeoHookeanMaterial (5e6, 0.45), false);
for (FemElement3d e : fem.getElements()) {
    // use element centroid to determine which elements are on the "spine"
    Point3d pos = new Point3d();
e.computeCentroid (pos);
    if (Math.abs(pos.y) <= 0.1+EPS) {
        bun.addElement (e);
    }
}
fem.addMaterialBundle (bun);

// add a control panel to control both the fem and bundle materials,
// as well as the fem and bundle widget sizes
ControlPanel panel = new ControlPanel();
panel.addWidget ("fem material", fem, "material");
panel.addWidget ("fem widget size", fem, "elementWidgetSize");
panel.addWidget ("bundle material", bun, "material");
panel.addWidget ("bundle widget size", bun, "elementWidgetSize");
addControlPanel (panel);

// set rendering properties, using element widgets
RenderProps.setFaceColor (fem, new Color (0.7f, 0.7f, 1.0f));
RenderProps.setFaceColor (bun, new Color (0.7f, 1.0f, 0.7f));
bun.setElementWidgetSize (0.9);
fem.setElementWidgetSize (0.8);

Lines 6-9 create a thin FEM hex sheet, centered on the origin, with size $1 \times 1 \times 0.1$ and $10 \times 10 \times 1$ elements along each axis. Its material is set to a linear material with a Young’s modulus of 10000. The leftmost nodes are then fixed (lines 11-16).

Next, a MuscleBundle is added and used to apply an additional material to the elements near the x axis (lines 19-29). It is given a neo-hookean material with a much higher stiffness, and the “spine” elements for it are selected by finding those whose centroids have a $y$ value within 0.1 of the x axis.

After the bundle is added, a control panel is created allowing interactive control of the materials for both the FEM model and the bundle, along with their elementWidgetSize properties.

Finally, some render properties are set (lines 41-44). The idea is to render the elements of both the FEM model and the bundle using element widgets (both of which will be visible since there is no surface rendering and the element widget sizes for both are greater than 0). The widget size for the bundle is made larger than that of the FEM model to ensure its widgets will cover those of the latter. Widget colors are controlled by the FEM and bundle face colors, set in lines 41-42.

The example can be run in ArtiSynth by selecting All demos > tutorial > MaterialBundleDemo from the Models menu. When it is run, the sheet will fall under gravity but be much stiffer along the spine (Figure 6.13). The control panel can be used to interactively adjust the material parameters for both the FEM and the bundle. This is one advantage of using bundles: the same material can be used to control multiple elements. The panel also allows interactive adjustment of the widget sizes, to illustrate what they look like and how they are rendered.

### 6.9 Muscle activated FEM models

Finite element muscle models are an extension to regular FEM models. As such, everything previously discussed for regular FEM models also applies to FEM muscles. Muscles have additional properties that allow them to contract when activated. There are two types of muscles supported:

**Fibre-based:** Point-to-point muscle fibres are embedded in the model.

**Material-based:** An auxiliary material is added to the constitutive law to embed muscle properties.

In this section, both types will be described.
6.9.1 FemMuscleModel

The main class for FEM-based muscles is FemMuscleModel, a subclass of FemModel3d. It differs from a basic FEM model in that it has the new property

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>muscleMaterial</td>
<td>An object that adds an activation-dependent ‘muscle’ term to the constitutive law.</td>
</tr>
</tbody>
</table>

This is a delegate object of type MuscleMaterial that computes activation-dependent stress and stiffness in the muscle. In addition to this property, FemMuscleModel adds two new lists of subcomponents:

- bundles
  - Groupings of muscle sub-units (fibres or elements) that can be activated.
- exciters
  - Components that control the activation of a set of bundles or other exciters.

6.9.1.1 Bundles

Muscle bundles allow for a muscle to be partitioned into separate groupings of fibres/elements, where each bundle can be activated independently. They are implemented in the class MuscleBundle. Bundles have three key properties:

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>excitation</td>
<td>Activation level of the muscle, $a \in [0,1]$.</td>
</tr>
<tr>
<td>fibresActive</td>
<td>Enable/disable “fibre-based” muscle components.</td>
</tr>
<tr>
<td>muscleMaterial</td>
<td>An object that adds an activation-dependent ‘muscle’ term to the constitutive law.</td>
</tr>
</tbody>
</table>

The excitation property controls the level of muscle activation, with zero being no muscle action, and one being fully activated. The fibresActive property is a boolean variable that controls whether or not to treat any contained fibres as point-to-point-like muscles (“fibre-based”). If false, the fibres are ignored. The third property, muscleMaterial, allows for a MuscleMaterial to be specified per bundle. By default, its value is inherited from FemMuscleModel.

Once a muscle bundle is created, muscle sub-units must be assigned to it. These are either point-to-point fibres, or material-based muscle element descriptors. The two types will be covered in Sections 6.9.2 and 6.9.3, respectively.

6.9.1.2 Exciters

Muscle exciters enable you to simultaneously activate a group of “excitation components”. This includes: point-to-point muscles, muscle bundles, muscle fibres, material-based muscle elements, and other muscle exciters. Components that can be excited all implement the ExcitationComponent interface. To add or remove a component to the exciter, use

```java
addTarget (ExcitationComponent ex);  // adds a component to the exciter
addTarget (ExcitationComponent ex, double gain);  // adds a component with a gain factor
removeTarget (ExcitationComponent ex);  // removes a component
```

If a gain factor is specified, the activation is scaled by the gain for that component.

6.9.2 Fibre-based muscles

In fibre-based muscles, a set of point-to-point muscle fibres are added between FEM nodes or markers. Each fibre is assigned an AxialMuscleMaterial, just like for regular point-to-point muscles (Section 4.4.1). Note that these muscle materials typically have a “rest length” property, that will likely need to be adjusted for each fibre. Once the set of fibres are added to a MuscleBundle, they need to be enabled. This is done by setting the fibresActive property of the bundle to true.

Fibres are added to a MuscleBundle using one of the functions:
addFibre( Muscle muscle ); // adds a point-to-point fibre
Muscle addFibre( Point p0, Point p1, 
  AxialMuscleMaterial mat); // creates and adds a fibre

The latter returns the newly created Muscle fibre. The following code snippet demonstrates how to create a fibre-based MuscleBundle and add it to an FEM muscle.

```java
// Create a muscle bundle
MuscleBundle bundle = new MuscleBundle("fibres");
Point3d[] fibrePoints = ... //create a sequential list of points

// Add fibres
Point pPrev = fem.addMarker(fibrePoints[0]); // create an FEM marker
for (int i=1; i<fibrePoints.length; i++) {
  Point pNext = fem.addMarker(fibrePoint[i]);

  // Create fibre material
  double l0 = pNext.distance(pPrev); // rest length
  AxialMuscleMaterial fibreMat =
    new BlemkerAxialMuscle(1.4*l0, l0, 3000, 0, 0);

  // Add a fibre between pPrev and pNext
  bundle.addFibre(pPrev, pNext, fibreMat); // add fibre to bundle
  pPrev = pNext;
}

// Enable use of fibres (default is disabled)
bundle.setFibresActive(true);

// Add the bundle to fem
fem.addMuscleBundle(bundle);
```

In these fibre-based muscles, force is only exerted between the anchor points of the fibres; it is a discrete approximation. These models are typically more stable than material-based ones.

### 6.9.3 Material-based muscles

In material-based muscles, the constitutive law is augmented with additional terms to account for muscle-specific properties. This is a continuous representation within the model.

The basic building block for a material-based muscle bundle is a `MuscleElementDesc`. This object contains a reference to a `FemElement3d`, a `MuscleMaterial`, and either a single direction or set of directions that specify the direction of contraction. If a single direction is specified, then it is assumed the entire element contracts in the same direction. Otherwise, a direction can be specified for each `integration point` within the element. A null direction signals that there is no muscle at the corresponding point. This allows for a sub-element resolution for muscle definitions. The positions of integration points for a given element can be obtained with:

```java
// loop through all integration points for a given element
for (IntegrationPoint3d ipnt : elem.getIntegrationPoints() ) {
  Point3d curPos = new Point3d();
  Point3d restPos = new Point3d();
  ipnt.computePosition (curPos, elem); // computes current position
  ipnt.computeRestPosition (restPos, elem); // computes rest position
}
```

By default, the `MuscleMaterial` is inherited from the bundle’s `material` property. Supported muscle materials include: `GenericMuscle`, `BlemkerMuscle`, and `FullBlemkerMuscle`. The Blemker-type materials are based on [3]. BlemkerMuscle only uses the muscle-specific terms (since a base material is provided the underlying FEM model), whereas FullBlemkerMuscle adds all terms described in the aforementioned paper.

Elements can be added to a muscle bundle using one of the methods:

```java
// Adds a muscle element
addElement (MuscleElementDesc elem);
```
The following snippet demonstrates how to create and add a material-based muscle bundle:

```java
// Create muscle bundle
MuscleBundle bundle = new MuscleBundle("embedded");

// Muscle material
MuscleMaterial muscleMat = new BlemkerMuscle(1.4, 1.0, 3000, 0, 0);
bundle.setMuscleMaterial(muscleMat);

// Muscle direction
Vector3d dir = Vector3d.X_UNIT;

// Add elements to bundle
for (FemElement3d elem : beam.getElements()) {
    bundle.addElement(elem, dir);
}

// Add bundle to model
beam.addMuscleBundle(bundle);
```

### 6.9.4 Example: comparison with two beam examples

Figure 6.14: FemMuscleBeams model loaded into ArtiSynth.

An example comparing a fibre-based and a material-based muscle is shown in Figure 6.14. The code can be found in artisynth.demos.tutorial.FemMuscleBeams. There are two FemMuscleModel beams in the model: one fibre-based, and one material-based. Each has three muscle bundles: one at the top (red), one in the middle (green), and one at the bottom (blue). In the figure, both muscles are fully activated. Note the deformed shape of the beams. In the fibre-based one, since forces only act between point on the fibres, the muscle seems to bulge. In the material-based muscle, the entire continuous volume contracts, leading to a uniform deformation.

Material-based muscles are more realistic. However, this often comes at the cost of stability. The added terms to the constitutive law are highly nonlinear, which may cause numerical issues as elements become highly contracted or highly deformed. Fibre-based muscles are, in general, more stable. However, they can lead to bulging and other deformation artifacts due to their discrete nature.
6.10 Fields

In modeling applications, particularly those employing FEM methods, situations often arise where it is necessary to describe numeric quantities that vary over some spatial domain. For example, the stiffness parameters of an FEM material may vary at different points within the volumetric mesh. When modeling muscles, the activation direction vector will also typically vary over the mesh.

ArtiSynth provides field components which can be used to represent such spatially varying quantities, together with mechanisms to attach these to certain properties within various materials. Field components can implement either scalar or vector fields. Scalar field components implement the interface `ScalarFieldComponent` and supply the method

```java
double getValue(Point3d p)
```

while vector fields implement `VectorFieldComponent` and supply the method

```java
T getValue(Point3d p)
```

where `T` parameterizes a class implementing `maspack.matrix.VectorObject`. In both cases, the idea is to provide values at arbitrary points over some spatial domain.

For vector fields, the `VectorObject` represented by `T` can generally be any fixed-size vector or matrix located in the package `maspack.matrix`, such as `Vector2d`, `Vector3d`, or `Matrix3d`. `T` can also be one of the variable-sized objects `VectorNd` or `MatrixNd`, although this requires using special wrapper classes and the sizing must remain constant within the field (Section 6.10.4).

Both `ScalarFieldComponent` and `VectorFieldComponent` are subclassed from `FieldComponent`. The reason for separating scalar and vector fields is simply efficiency: having scalar fields work with the primitive type `double`, instead of the object `Double`, requires considerably less storage and somewhat less computational effort.

A field is typically implemented by specifying a finite set of values at discrete locations on an underlying spatial grid and then using an interpolation method to determine values at arbitrary points. The field components themselves are defined within the package `artisynth.core.fields`.

At present, three types of fields are implemented:

Grid fields

- The most basic type of field, in which the values are specified at the vertices of a regular Cartesian grid and then interpolated between vertices. As discussed further in Section 6.10.1, there are two main types grid field component: `ScalarGridField` and `VectorGridField<T>`.

FEM fields

- Fields for which values are specified at the features of an FEM mesh (e.g., nodes, elements or element integration points). As discussed further in Section 6.10.2, there are six primary types of FEM field component: `ScalarNodalField` and `VectorNodalField<T>` (nodes), `ScalarElementField` and `VectorElementField<T>` (elements), and `ScalarSubElemField` and `VectorSubElemField<T>` (integration points).

Mesh fields

- Fields in which values are specified for either the vertices or faces of a triangular mesh. As discussed further in Section 6.10.3, there are four primary types of mesh field component: `ScalarVertexField` and `VectorVertexField<T>` (vertices), and `ScalarFaceField` and `VectorFaceField<T>` (faces).

Within an application model, field deployment typically involves the following steps:

1. Create the field component and add it to the model. Grid and mesh fields are usually added to the fields component list of a `MechModel`, while FEM fields are added to the fields list of the `FemModel3d` for which the field is defined.

2. Specify field values for the appropriate features (e.g., grid vertices, FEM nodes, mesh faces).

3. If necessary, bind any needed material properties to the field, as discussed further in Section 6.10.5.
As a simple example of the first two steps, the following code fragment constructs a scalar nodal field associated with an FEM model named `fem`:

```java
FemModel3d fem;
// ... build the fem ...
ScalarNodalField field = new ScalarNodalField("stiffness", fem, 0);
for (FemNode3d n : fem.getNodes()) {
    double value = ... // compute value for the node
    field.setValue(n, value);
}
fem.addField(field); // add the field to the fem
```

More details on specific field components are given in the sections below.

### 6.10.1 Grid fields

A grid field specifies scalar or vector values at the vertices of a regular Cartesian grid and interpolates values between vertices. ArtiSynth currently provides two grid field components in `artisynth.core.fields`:

- `ScalarGridField`
- `VectorGridField<T>`

where T is any class implementing `maspack.matrix.VectorObject`. For both grid types, the value returned by `getValue(p)` is determined by finding the grid cell containing p and then using trilinear interpolation of the surrounding nodal values. If p lies outside the grid volume, then `getValue(p)` either returns the value at the nearest grid point p' (if the component property `clipToGrid` is set to true), or else returns a special value indicating that p is outside the grid. This special value is `ScalarGridField.OUTSIDE_GRID` for scalar fields or `null` for vector fields.

Grid field components are implemented as wrappers around the more basic objects `ScalarGrid` and `VectorGrid<T>` defined in `maspack.geometry`. Applications first create one of these primary grid objects and then use it to create the field component. Instances of `ScalarGrid` and `VectorGrid<T>` can be created with constructors such as

```java
ScalarGrid (Vector3d widths, Vector3i res)
ScalarGrid (Vector3d widths, Vector3i res, RigidTransform3d TCL)
VectorGrid (Class<T> type, Vector3d widths, Vector3i res)
VectorGrid (Class<T> type, Vector3d widths, Vector3i res, RigidTransform3d TCL)
```

where `widths` gives the grid widths along each of the x, y and z axes, `res` gives the number of cells along each axis, and for vector grids `type` is the class type of the `maspack.matrix.VectorObject` object parameterized by T. TCL is an optional argument which, if not null, describes the position and orientation of the grid center with respect to local coordinates; otherwise, in local coordinates the grid is centered at the origin and aligned with the x, y and z axes.

For the vector grid constructors above, T should be a fixed-size vector or matrix, such as `Vector3d` or `Matrix3d`. The variable sized objects `VectorNd` and `MatrixNd` can also be used with the aid of special wrapper classes, as described in Section 6.10.4.

By default, grids are axis-aligned and centered at the origin of the world coordinate system. A transform `TLW` can be specified to place the grid at a different position and/or orientation. `TLW` represents the transform from local to world coordinates and can be controlled with the methods

```java
void setLocalToWorld (RigidTransform3d TLW)
RigidTransform3d getLocalToWorld()
```

If not specified, `TLW` is the identity and local and world coordinates are the same.

Once a grid is created, it can be used to instantiate a grid field component using one of the constructors

```java
ScalarGridField (ScalarGrid grid)
ScalarGridField (String name, ScalarGrid grid)
VectorGridField (VectorGrid grid)
VectorGridField (String name, VectorGrid grid)
```
where grid is the primary grid and name is an optional component name. Note that the primary grid is not copied, so any subsequent changes to it will be reflected in the enclosing field component.

Once the grid field component is created, its values can be set by specifying the values at its vertices. The methods to query and set vertex values for scalar and vector fields are

```java
int numVertices()

// scalar fields:
double getVertexValue (int xi, int yj, int zk)
double getVertexValue (int vi)
void setVertexValue (int xi, int yj, int zk, double value)
void setVertexValue (int vi, double value)

// vector fields:
T getVertexValue (int xi, int yj, int zk)
T getVertexValue (int vi)
void setVertexValue (int xi, int yj, int zk, T value)
void setVertexValue (int vi, T value)
```

where xi, yj, and zk are the vertex’s indices along the x, y, and z axes, and vi is a general index that should be in the range 0 to field.numVertices()-1 and is related to xi, yj, and zk by

\[ vi = xi + nx*yj + (nx*ny)*zk, \]

where nx and ny are the number of vertices along the x and y axes.

When computing a grid value using getValue(p), the point p is assumed to be in either grid local or world coordinates, depending on whether the field component’s property localValuesForField is true or false (local and world coordinates are the same unless the primary grid’s local-to-world transform TLW has been set as described above).

To find the spatial position of a vertex within a grid field component, one may use the methods

```java
Vector3d getVertexPosition (xi, yj, zk)
Vector3d getVertexPosition (vi)
```

which return the vertex position in either local or world coordinates depending on the setting of localValuesForField.

The following code example shows the creation of a ScalarGridField, with widths 5 \times 5 \times 10 and a cell resolution of 10 \times 10 \times 20, centered at the origin and whose vertex values are set to their distance from the origin, in order to create a simple distance field:

```java
// create a 5 x 5 x 10 grid with resolution 10 x 10 x 20 centered on the // origin
ScalarGrid grid = new ScalarGrid (new Vector3d (5, 5, 10), new Vector3i (10, 10, 20));

// use this to create a scalar grid field where the value at each vertex // is the distance from the origin
ScalarGridField field = new ScalarGridField (grid);

// iterate through all the vertices and set the value for each to its // distance from the origin, as given by the norm of it’s position for (int vi=0; vi<grid.numVertices(); vi++) {
    Vector3d vpos = grid.getVertexCoords (vi);
    grid.setVertexValue (vi, vpos.norm());
}
// add to a mech model:
mech.addField (field);
```

As shown in the example and as mentioned earlier, grid field are generally added to the fields list of a MechModel.
6.10.2  FEM fields

A FEM field specifies scalar or vector values for the features of an FEM mesh. These can be either the nodes (nodal fields), elements (element fields), or element integration points (sub-element fields). As such, FEM fields provide a way to augment the mesh to store application-specific quantities, and are typically used to bind properties of FEM materials that need to vary over the domain (Section 6.10.5).

To evaluate getValue(p) at an arbitrary point p, the field finds the FEM element containing p (or nearest to p, if p is outside the FEM mesh), and either interpolates the value from the surrounding nodes (nodal fields), uses the element value directly (element fields), or interpolates from the integration point values (sub-element fields).

When finding a FEM field value at a point p, getValue(p) is evaluated with respect to the FEM model’s current spatial position, as opposed to its rest position.

FEM fields maintain a default value which describes the value at features for which values are not explicitly set. If unspecified, the default value itself is zero.

In the remainder of this section, it is assumed that vector fields are constructed using fixed-size vectors or matrices, such as Vector3d or Matrix3d. However, it is also possible to construct fields using VectorNd or MatrixNd, with the aid of special wrapper classes, as described in Section 6.10.4. That section also details a convenience wrapper class for Vector3d.

The three field types are now described in detail.

6.10.2.1  Nodal fields

Implemented by ScalarNodalField, VectorNodalField<T>, or subclasses of the latter, nodal fields specify their values at the nodes of an FEM mesh. They can be created with constructors such as

```java
// scalar fields:
ScalarNodalField (FemModel3d fem)
ScalarNodalField (FemModel3d fem, double defaultValue)
ScalarNodalField (String name, FemModel3d fem, double defaultValue)

// vector fields (with vector type parameterized by T):
VectorNodalField (Class<T> type, FemModel3d fem)
VectorNodalField (Class<T> type, (FemModel3d fem, T defaultValue)
VectorNodalField (String name, Class<T> type, FemModel3d fem, T defaultValue)
```

where fem is the associated FEM model, defaultValue is the default value, and name is a component name. For vector fields, the maspack.matrix.VectorObject type is parameterized by T, and type gives its actual class type (e.g., Vector3d.class).

Once the grid has been created, values can be queried and set at the nodes using methods such as

```java
// scalar fields:
void setValue (FemNode3d node, double value) // set value for node
double getValue (FemNode3d node) // get value for node
double getValue (int nodeNum) // get value using node number

// vector fields:
void setValue (FemNode3d node, T value) // set value for node
T getValue (FemNode3d node) // get value for node
T getValue (int nodeNum) // get value using node number

// all fields:
boolean isValueSet (FemNode3d node) // query if value set for node
void clearValue (FemNode3d node) // unset value for node
void clearAllValues () // unset values for all nodes
```

Here nodeNum refers to an FEM node’s number (which can be obtained using node.getNumber()). Numbers are used instead of indices to identity FEM nodes and elements because they are guaranteed to be persistent in case of mesh
editing, and will remain unchanged as long as the node or element is not removed from the FEM model. Note that values don’t need to be set at all nodes, and set values can be unset using the clear methods. For nodes with no value set, the getValue() methods will return the default value.

As a simple example, the following code fragment constructs a ScalarNodalField for an FEM model fem that describes stiffness values at every node of an FEM:

```java
// instantiate the field and set values for each node:
ScalarNodalField field = new ScalarNodalField("stiffness", fem);
for (FemNode3d n: fem.getNodes()) {
    double stiffness = ... // compute stiffness value for the node
    field.setValue(n, stiffness);
}
fem.addField(field); // add the field to the fem
```

As noted earlier, FEM fields should be stored in the fields list of the associated FEM model.

Another example shows the creation of a field of 3D direction vectors:

```java
VectorNodalField<Vector3d> field =
    new VectorNodalField<Vector3d> ("directions", Vector3d.class, fem);
Vector3d dir = new Vector3d();
for (FemNode3d node : fem.getNodes()) {
    ... // compute direction and store in dir
    field.setValue(node, dir);
}
```

When creating a vector field, the constructor needs the class type of the field’s VectorObject. However, for the special case of Vector3d, one may also use the convenience wrapper class Vector3dNodalField, described in Section 6.10.4.

### 6.10.2.2 Element fields

Implemented by ScalarElementField, VectorElementField<T>, or subclasses of the latter, element fields specify their values at the elements of an FEM mesh, and these values are assumed to be constant within the element. To evaluate getValue(p), the field finds the containing (or nearest) element for p, and then returns the value for that element.

Because values are assume to be constant within each element, element fields are inherently discontinuous across element boundaries.

The constructors for element fields are analogous to those for nodal fields:

```java
// scalar fields:
ScalarElementField (FemModel3d fem)
ScalarElementField (FemModel3d fem, double defaultValue)
ScalarElementField (String name, FemModel3d fem, double defaultValue)

// vector fields (with vector type parameterized by T):
VectorElementField (Class<T> type, FemModel3d fem)
VectorElementField (Class<T> type, (FemModel3d fem, T defaultValue)
VectorElementField (String name, Class<T> type, FemModel3d fem, T defaultValue)
```

and the methods for setting values at elements are similar as well:

```java
// scalar fields:
void setValue (FemElement3dBase elem, double value) // set value for element
double getValue (FemElement3dBase elem) // get value for element
double getElementValue (int elemNum) // get value for volume element
double getShellElementValue (int elemNum) // get value for shell element

// vector fields:
void setValue (FemElement3dBase elem, T value) // set value for element
T getValue (FemElement3dBase elem) // get value for element
T getElementValue (int elemNum) // get value for volume element
```
The elements associated with an element field can be either volume or shell elements, which are stored in the FEM component lists `elements` and `shellElements`, respectively. Since volume and shell elements may share element numbers, the separate methods `getElementValue()` and `getShellElementValue()` are used to access values by element number.

The following code fragment constructs a `VectorElementField` based on `Vector2d` that stores a 2D coordinate value at all regular and shell elements:

```java
VectorElementField<Vector2d> efield = new VectorElementField<Vector2d> ("coordinates", Vector2d.class, fem);
Vector2d coords = new Vector2d();
for (FemElement3d e : fem.getElements()) {
  ... // compute coordinate values and store in coords
  efield.setValue (e, coords);
}
for (ShellElement3d e : fem.getShellElements()) {
  ... // compute coordinate values and store in coords
  efield.setValue (e, coords);
}
```

### 6.10.2.3 Sub-element fields

Implemented by `ScalarSubElemField`, `VectorSubElemField<T>`, or subclasses of the latter, sub-element fields specify their values at the integration points within each element of an FEM mesh. These fields are used when we need precisely computed information at each of the element’s integration points, and we can’t assume that nodal interpolation will give an accurate enough approximation.

To evaluate `getValue(p)`, the field finds the containing (or nearest) element for `p`, extrapolating the integration point values back to the nodes, and then using nodal interpolation.

Constructors are similar to those for element fields:

```java
// scalar fields:
ScalarSubElemField (FemModel3d fem)
ScalarSubElemField (FemModel3d fem, double defaultValue)
ScalarSubElemField (String name, FemModel3d fem, double defaultValue)

// vector fields:
VectorSubElemField (Class<T> type, FemModel3d fem)
VectorSubElemField (Class<T> type, FemModel3d fem, T defaultValue)
VectorSubElemField (String name, Class<T> type, FemModel3d fem, T defaultValue)
```

Value accessors are also similar, except that an additional argument `k` is required to specify the index of the integration point:

```java
// scalar fields:
void setValue (FemElement3dBase e, int k, double value) // set value for point
double getValue (int elemNum, int k) // get value for point
double getElementValue (FemElement3dBase e, int k) // get value for volume point
double getShellElementValue (int elemNum, int k) // get value for shell point

// vector fields:
void setValue (FemElement3dBase e, T value, int k) // set value for point
T getValue (FemElement3dBase e, int k) // get value for point
T getElementValue (int elemNum, int k) // get value for volume point
T getShellElementValue (int elemNum, int k) // get value for shell point
```
The integration point index \( k \) should be in the range 0 to \( n - 1 \), where \( n \) is the total number of integration points for the element in question and can be queried by the method `numAllIntegrationPoints()`.

The total number of integration points \( n \) includes both the regular integration point as well as the warping point, which is located at the element center, is indexed by \( n - 1 \), and is used for corotated linear materials. If a `SubElemField` is being used to supply mesh-varying values to one of the linear material parameters (Section 6.10.5), then it is important to supply values at the warping point.

The example below shows the construction of a `ScalarSubElemField`:

```java
ScalarSubElemField field = new ScalarSubElemField("modulus", fem);
for (FemElement3d e : fem.getElements()) {
    IntegrationPoint3d[] ipnts = e.getAllIntegrationPoints();
    for (int k=0; k<ipnts.length; k++) {
        double value = ... // compute value at integration point k
        field.setValue(e, k, value);
    }
}
fem.addField(field); // add the field to the fem
```

First, the field is instantiated with the name "modulus". The code then iterates first through the FEM elements, and then through each element’s integration points, computing values appropriate to each one. A list of each element’s integration points is returned by `e.getAllIntegrationPoints()`. Having access to the actual integration points is useful in case information about them is needed to compute the field values. In particular, if it is necessary to obtain an integration point’s rest or current (spatial) position, these can be queried as follows:

```java
IntegrationPoint3d ipnt;
Point3d pos = new Point3d();
FemElement3d elem;
...

// compute spatial position:
ipnt.computePosition(pos, elem.getNodes());

// compute rest position:
ipnt.computeRestPosition(pos, elem.getNodes());
```

The regular integration points, excluding the warping point, can be queried using `getIntegrationPoints()` and `numIntegrationPoints()` instead of `getAllIntegrationPoints()` and `numAllIntegrationPoints()`.

### 6.10.3 Mesh fields

A mesh field specifies scalar or vector values for the features of an FEM mesh. These can be either the vertices (vertex fields) or faces (face fields). Mesh fields have been introduced in particular to allow properties of contact force behaviors (Section 7.7.2), such as `LinearElasticContact`, to vary over the contact mesh. Mesh fields can currently be defined only for triangular polyhedral meshes.

To evaluate `getValue(p)` at an arbitrary point \( p \), the field finds the mesh face nearest to \( p \), and then either interpolates the value from the surrounding vertices (vertex fields) or uses the face value directly (face fields).

Mesh fields make use of the classes `PolygonalMesh`, `Vertex3d`, and `Face`, defined in the package `maspack.geometry`. They maintain a default value which describes the value at features for which values are not explicitly set. If unspecified, the default value itself is zero.
In the remainder of this section, it is assumed that vector fields are constructed using fixed-size vectors or matrices, such as `Vector3d` or `Matrix3d`. However, it is also possible to construct fields using `VectorNd` or `MatrixNd`, with the aid of special wrapper classes, as described in Section 6.10.4. That section also details a convenience wrapper class for `Vector3d`.

The two field types are now described in detail.

### 6.10.3.1 Vertex fields

Implemented by `ScalarVertexField`, `VectorVertexField<T>`, or subclasses of the latter, vertex fields specify their values at the vertices of a mesh. They can be created with constructors such as

```java
// scalar fields:
ScalarVertexField (MeshComponent mcomp)
ScalarVertexField (MeshComponent mcomp, double defaultValue)
ScalarVertexField (String name, MeshComponent mcomp, double defaultValue)

// vector fields (with vector type parameterized by T):
VectorVertexField (Class<T> type, MeshComponent mcomp)
VectorVertexField (Class<T> type, (MeshComponent mcomp, T defaultValue)
VectorVertexField (String name, Class<T> type, MeshComponent mcomp, T defaultValue)
```

where `mcomp` is a mesh component containing the mesh, `defaultValue` is the default value, and `name` is a component name. For vector fields, the `maspack.matrix.VectorObject` type is parameterized by `T`, and `type` gives its actual class type (e.g., `Vector3d.class`).

Once the field has been created, values can be queried and set at the vertices using methods such as

```java
// scalar fields:
void setValue (Vertex3d vtx, double value)  // set value for vertex
double getValue (Vertex3d vtx)           // get value for vertex
double getValue (int vidx)              // get value for vertex number

// vector fields:
void setValue (Vertex3d vtx, T value)     // set value for vertex
T getValue (Vertex3d vtx)                // get value for vertex
T getValue (int vidx)                    // get value for vertex number

// all fields:
boolean isValueSet (Vertex3d vtx)        // query if value set for vertex
void clearValue (Vertex3d vtx)           // unset value for vertex
void clearAllValues()                    // unset values for all vertices
```

where `vidx` is the vertex’s index. Note that values don’t need to be set at all vertices, and set values can be unset using the `clear` methods. For vertices with no value set, `getValue()` will return the default value.

As a simple example, the following code fragment constructs a `ScalarVertexField` for a mesh contained in the `MeshComponent mcomp` that describes contact stiffness values at every vertex:

```java
MechModel mech; // containing mech model
...

// instantiate the field and set values for each node:
ScalarVertexField field = new ScalarVertexField("stiffness", mcomp);
// extract the mesh from the component
PolygonalMesh mesh = (PolygonalMesh)mcomp.getMesh();
for (Vertex3d vtx : mesh.getVertices()) {
    double stiffness = ...  // compute stiffness value for the vertex
    field.setValue (vtx, stiffness);
}
mech.addField (field);  // add the field to the mech model
```
As noted earlier, mesh fields are usually stored in the fields list of a MechModel.

Another example shows the creation of a field of 3D direction vectors:

```java
VectorVertexField<Vector3d> field =
    new VectorVertexField<Vector3d> ("directions", Vector3d.class, mcomp);
Vector3d dir = new Vector3d();
for (Vertex3d vtx : mesh.getVertices()) {
    // compute direction and store in dir
    field.setValue(vtx, dir);
}
```

When creating a vector field, the constructor needs the class type of the field’s VectorObject. However, for the special case of Vector3d, one may also use the convenience wrapper class Vector3dVertexField, described in Section 6.10.4.

6.10.3.2 Face fields

Implemented by ScalarFaceField, VectorFaceField<T>, or subclasses of the latter, face fields specify their values at the faces of a polygonal mesh, and these values are assumed to be constant within the face. To evaluate getValue(p), the field finds the containing (or nearest) face for p, and then returns the value for that face.

Because values are assumed to be constant within each face, face fields are inherently discontinuous across face boundaries.

The constructors for face fields are analogous to those for vertex fields:

```java
// scalar fields:
ScalarFaceField (MeshComponent mcomp)
ScalarFaceField (MeshComponent mcomp, double defaultValue)
ScalarFaceField (String name, MeshComponent mcomp, double defaultValue)

// vector fields (with vector type parameterized by T):
VectorFaceField (Class<T> type, MeshComponent mcomp)
VectorFaceField (Class<T> type, (MeshComponent mcomp, T defaultValue)
VectorFaceField (String name, Class<T> type, MeshComponent mcomp, T defaultValue)
```

and the methods for setting values at faces are similar as well, where fidx refers to the face index:

```java
void setValue (Face face, double value)       // set value for face
double getValue (Face face)                   // get value for face
double getValue (int fidx)                    // get value using face index

// vector fields:
void setValue (Face face, T value)            // set value for face
T getValue (Face face)                        // get value for face
T getValue (int fidx)                         // get value using face index

// all fields:
boolean isValueSet (Face face)               // query if value set for face
void clearValue (Face face)                  // unset value for face
void clearAllValues ()                       // unset values for all faces
```

The following code fragment constructs a VectorFaceField based on Vector2d that stores a 2D coordinate value at all faces of a mesh:

```java
VectorFaceField<Vector2d> field =
    new VectorFaceField<Vector2d> ("coordinates", Vector2d.class, mcomp);
Vector2d coords = new Vector2d();
// extract the mesh from the component
PolygonalMesh mesh = (PolygonalMesh)mcomp.getMesh();
for (Face face : mesh.getFaces()) {
    ... // compute coordinate and store in coord
    field.setValue(face, coords);
}
```
6.10.4 Fields for VectorNd, MatrixNd and Vector3d

The vector fields described above can be implemented for most fixed-size vectors and matrices defined in the package maspack.matrix (e.g., Vector2d, Vector3d, Matrix3d). However, if vectors or matrices with different size are needed, it is also possible to create vector fields using VectorNd and MatrixNd, provided that the sizing remain constant within any given field. This is acheived using special wrapper classes that contain the sizing information, which is supplied in the constructors. For convenience, wrapper classes are also provided for vector fields that use Vector3d, since that vector size is quite common.

For vector FEM and mesh fields, the complete set of wrapper classes is listed below:

<table>
<thead>
<tr>
<th>Class</th>
<th>base class</th>
<th>vector type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vector3dNodalField</td>
<td>VectorNodalField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdNodalField</td>
<td>VectorNodalField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdNodalField</td>
<td>VectorNodalField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
<tr>
<td>Vector3dElementField</td>
<td>VectorElementField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdElementField</td>
<td>VectorElementField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdElementField</td>
<td>VectorElementField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
<tr>
<td>Vector3dSubElemField</td>
<td>VectorSubElemField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdSubElemField</td>
<td>VectorSubElemField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdSubElemField</td>
<td>VectorSubElemField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
<tr>
<td>Vector3dVertexField</td>
<td>VectorVertexField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdVertexField</td>
<td>VectorVertexField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdVertexField</td>
<td>VectorVertexField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
<tr>
<td>Vector3dFaceField</td>
<td>VectorFaceField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdFaceField</td>
<td>VectorFaceField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdFaceField</td>
<td>VectorFaceField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
<tr>
<td>Vector3dSubElemField</td>
<td>VectorSubElemField&lt;T&gt; Vector3d</td>
<td></td>
</tr>
<tr>
<td>VectorNdSubElemField</td>
<td>VectorSubElemField&lt;T&gt; VectorNd</td>
<td></td>
</tr>
<tr>
<td>MatrixNdSubElemField</td>
<td>VectorSubElemField&lt;T&gt; MatrixNd</td>
<td></td>
</tr>
</tbody>
</table>

These all behave identically to their base classes, except that their constructors omit the type argument (since this is built into the class definition) and, for VectorNd and MatrixNd, supply information about the vector or matrix sizes. For example, constructors for the FEM nodal fields include

- Vector3dNodalField (FemModel3d fem)
- Vector3dNodalField (FemModel3d fem, Vector3d defaultValue)
- Vector3dNodalField (String name, FemModel3d fem, Vector3d defaultValue)
- VectorNdNodalField (int vecSize, FemModel3d fem)
- VectorNdNodalField (int vecSize, FemModel3d fem, VectorNd defaultValue)
- VectorNdNodalField (int vecSize, String name, FemModel3d fem, VectorNd defaultValue)
- MatrixNdNodalField (int rowSize, int colSize, FemModel3d fem)
- MatrixNdNodalField (int rowSize, int colSize, FemModel3d fem, MatrixNd defaultValue)
- MatrixNdNodalField (int rowSize, int colSize, String name, FemModel3d fem, MatrixNd defaultValue)

where vecSize, rowSize and colSize give the sizes of the VectorNd or MatrixNd objects within the field. Constructors for other field types are equivalent and are described in their API documentation.

For grid fields, one can use either VectorNdGrid or MatrixNdGrid, both defined in maspack.geometry, as the primary grid used to construct the field. Constructors for these include

- VectorNdGrid (int VecSize, Vector3d widths, Vector3i res)
- MatrixNdGrid (int rowSize, int colSize, Vector3d widths, Vector3i res, RigidTransform3d TCL)
6.10.5  Binding material properties

A principal purpose of field components is to enable certain FEM material properties to vary spatially over the FEM geometry. Many material properties can be bound to a field, so that when they are queried internally by the solver at the integration points for each FEM element, the field value at that integration point is used instead of the regular property value. Likewise, some properties of contact force behaviors, such as the Young's Modulus and thickness properties of LinearElasticContact (Section 7.7.3) can be bound to a field that varies over the contact mesh geometry.

To bind a property to a field, it is necessary that

1. The type of the field matches the value of the property;
2. The property is itself bindable.

If the property has a double value, then it can be bound to any ScalarFieldComponent. Otherwise, if the property has a value $T$, where $T$ is an instance of maspack.matrix.VectorObject, then it can be bound to a vector field.

Bindable properties export two methods with the signature

```java
F setXXXField (F field)
F getXXXField ()
```

where XXX is the property name and $F$ is an appropriate field type.

As long as a property XXX is bound to a field, its regular value will still appear (and can be set) through widgets in control or property panels, or via its getXXX() and setXXX() accessors. However, the regular value won’t be used internally by the FEM simulation.

For example, consider the Young's Modulus property, which is present in several FEM materials, including LinearMaterial and NeoHookeanMaterial. This has a double value, and is bindable, and so can be bound to a ScalarFieldComponent as follows:

```java
FemModel3d fem;
ScalarNodalField field;
NeoHookeanMaterial mat;
...
other initialization ...
mat.setYoungsModulusField (field); // bind to the field
fem.setMaterial (mat); // set the material in the FEM model
```

It is important to perform field bindings on materials before they are set in an FEM model (or one of its subcomponents, such as MuscleBundles). That’s because the setMaterial() method copies the input material, and so any settings made on it afterwards won’t be seen by the FEM:

```java
// works: field will be seen by the copied version of 'mat'
mant.setYoungsModulusField (field);
fem.setMaterial (mat);

// does NOT work: field not seen by the copied version of 'mat'
fem.setMaterial (mat);
mant.setYoungsModulusField (field);
```

To unbind YoungsModulus, one can call

```java
mat.setYoungsModulusField (null);
```
Usually, FEM fields are used to bind properties in FEM materials while mesh fields are used for contact properties, but other fields can be used, particularly grid fields. To understand this a bit better, we discuss briefly some of the internals of how binding works. When evaluating stresses, FEM materials have access to a FemFieldPoint object that provides information about the integration point, including its element, nodal weights, and integration point index. This can be used to rapidly evaluate values within nodal, element and sub-element FEM fields. Likewise, when evaluating contact forces, contact materials like LinearElasticContact have access to a MeshFieldPoint object that describes the contact point position as a weighted sum of mesh vertex positions, which can then be used to rapidly evaluate evaluate values within vertex or face mesh fields. However, all fields, regardless of type, implement methods for determining values at both FemFieldPoints and MeshFieldPoints:

```cpp
T getValue(FemFieldPoint fp)
T getValue(MeshFieldPoint mp)
```

If necessary, these methods simply fall back on the getValue(Point3d) method, which is defined for all fields and works relatively fast for grid fields.

There are some additional details to consider when binding FEM material properties:

1. When binding to a grid field, one has the choice of whether to use the grid to represent values with respect to the FEM’s rest position or spatial position. This can be controlled by setting the useFemRestPositions property of the grid field, the default value for which is true.

2. When binding the bulkModulus property of an incompressible material, it is best to use a nodal field if the FEM model is using nodal-based soft incompressibility (i.e., the model’s softIncompMethod property is set to either NODAL or AUTO; Section 6.7.3). That’s because soft nodal incompressibility require evaluating the bulk modulus property at nodes instead of integration points, which is much easier to do with a nodal-based field.

### 6.10.6 Example: FEM with variable stiffness

![VariableStiffness model](image)

Figure 6.15: VariableStiffness model after being run in ArtiSynth.

A simple model demonstrating a stiffness that varies over an FEM mesh is defined in

`artisynth.demos.tutorial.VariableStiffness`

It consists of a simple thin hexahedral beam with a linear material for which the Young’s modulus \(E\) is made to vary nonlinearly along the x axis of the rest position according to the formula

\[
E = \frac{10^8}{1 + 1000 x^3}
\]

(6.10)

The model’s build method is given below:
public void build (String[] args) {
    MechModel mech = new MechModel (*mech*);
    addModel (mech);

    // create regular hex grid FEM model
    FemModel3d fem = FemFactory.createHexGrid {
        null, 1.0, 0.25, 0.05, 20, 5, 1);
    fem.transformGeometry (new RigidTransform3d (0.5, 0, 0)); // shift right
    fem.setDensity (1000.0);
    mech.addModel (fem);

    // fix the left-most nodes
    double EPS = 1e-8;
    for (FemNode3d n : fem.getNodes()) {
        if (n.getPosition().x < EPS) {
            n.setDynamic (false);
        }
    }

    // create a scalar nodel field to make the stiffness vary
    // nonlinearly along the rest position x axis
    ScalarNodalField stiffnessField = new ScalarNodalField(fem, 0);
    for (FemNode3d n : fem.getNodes()) {
        double s = 10*(n.getRestPosition().x);
        double E = 100000000*(1/(1+s*s*s));
        stiffnessField.setValue (n, E);
    }
    fem.addField (stiffnessField);

    // create a linear material, bind its Young's modulus property to the
    // field, and set the material in the FEM model
    LinearMaterial linearMat = new LinearMaterial (100000, 0.49);
    linearMat.setYoungsModulusField (stiffnessField); //, /* useRestPos=*/true);
    fem.setMaterial (linearMat);

    // set some render properties for the FEM model
    fem.setSurfaceRendering (SurfaceRender.Shaded);
    RenderProps.setFaceColor (fem, Color.CYAN);
}

Lines 6-10 create the hex FEM model and shift it so that the left side is aligned with the origin, while lines 12-17 fix
the leftmost nodes. Lines 21-27 create a scalar nodal field for the Young's modulus, with lines 23-24 computing $E$
according to (6.10). The field is then bound to a linear material which is then set in the model (lines 30-32).

The example can be run in ArtiSynth by selecting All demos > tutorial > VariableStiffness from the Models menu. When
run, the beam will bend under gravity, but mostly on the right side, due to the much higher stiffness on the left (Figure 6.15).

6.10.7 Example: specifying FEM muscle directions

Another example involves using a Vector3d field to specify the muscle activation directions over an FEM model and is
defined in

artisynth.demos.tutorial.RadialMuscle

When muscles are added using MuscleBundles (Section 6.9), the muscle directions are stored and handled internally
by the muscle bundle itself. However, it is possible to add a MuscleMaterial directly to the elements of an FEM model,
using a MaterialBundle (Section 6.8), in which case the directions need to be set explicitly using a field.

The model's build method is given below:

public void build (String[] args) {
    MechModel mech = new MechModel (*mech*);
    addModel (mech);
Figure 6.16: (Top) RadialMuscle model after being loaded into ArtiSynth and run with its excitation set to 0. (Bottom) RadialMuscle with its excitation set to around .375.

```java
// create a thin cylindrical FEM model with two layers along z
double radius = 0.8;
FemMuscleModel fem = new FemMuscleModel("radialMuscle");
mech.addModel(fem);
fem.setDensity(1000);
FemFactory.createCylinder(fem, radius/8, radius, 20, 2, 8);
fem.setMaterial(new NeoHookeanMaterial(200000.0, 0.33));
// fix the nodes close to the center
for (FemNode3d node : fem.getNodes()) {
    Point3d pos = node.getPosition();
    double radialDist = Math.sqrt(pos.x*pos.x + pos.y*pos.y);
    if (radialDist < radius/2) {
        node.setDynamic(false);
    }
}
// compute a direction field, with the directions arranged radially
Vector3d dir = new Vector3d();
Vector3dElementField dirField = new Vector3dElementField (fem);
for (FemElement3d elem : fem.getElements()) {
    elem.computeCentroid(dir);
    // set directions only for the upper layer elements
    if (dir.z > 0) {
```
Lines 5-19 create a thin cylindrical FEM model, centered on the origin, with radius \( r \) and height \( r/8 \), consisting of hexes with wedges at the center, with two layers of elements along the \( z \) axis (which is parallel to the cylinder axis). Its base material is set to a neo-hookean material. To keep the model from falling under gravity, all nodes whose distance to the \( z \) axis is less than \( r/2 \) are fixed.

Next, a Vector3d field is created to specify the directions, on a per-element basis, for the muscle material which will be added subsequently (lines 21-32). While we could create an instance of `VectorElementField<Vector3d>`, we use `Vector3dElementField`, since this is available and provides additional functionality (such as the ability to render the directions). Directions are set to lie outward in a radial direction perpendicular to the \( z \) axis, and since the model is centered on the origin, they can be computed easily by first computing the element centroids, removing the \( z \) component, and then normalizing. In order to give the muscle action an upward bias, we only set directions for elements in the upper layer. Direction values for elements in the lower layer will then automatically have a default value of 0, which will cause the muscle material to not apply any stress.

We next add to the model a muscle material whose directions will be determined by the field. To hold the material, we first create and add a `MaterialBundle` which is set to act on all elements (line 35-36). Then we set this bundle’s material to `SimpleForceMuscle`, which adds a stress along the muscle direction that equals the excitation value times the value of its `maxStress` property, and bind the material’s `restDir` property to the direction field (lines 37-39).

Finally, we create and add a control panel to allow interactive control over the muscle material’s excitation property (lines 42-44), and set some rendering properties for the FEM model.

The example can be run in ArtiSynth by selecting `All demos > tutorial > RadialMuscle` from the Models menu. When it is first run, it falls around the edges under gravity (Figure 6.16, top). Applying an excitation causes a radial contraction which pulls the edges upward and, if high enough, causes them to buckle (Figure 6.16, bottom).

### 6.11 Rendering and Visualizations

In addition to the standard `RenderProps` that control how the nodes and surfaces appear, finite element models and their subcomponents have a few additional properties that affect rendering. Some of these are listed in Table 6.6.

The property `elementWidgetSize` applies only to `FemModel3d` and `FemElement3d`. It specifies the scale to draw each element volume. For instance, the blue beam in Figure 7.2 uses a widget size of 0.8, resulting in a mosaic-like pattern.

The next two properties in Table 6.6 apply to the muscle classes `FemMuscleModel`, `MuscleBundle`, and `MuscleElementDesc`. When `directionRenderLen > 0`, lines are drawn inside elements to indicate fibre directions. If `directionRenderType = ELEMENT`, then one line is drawn per element indicating the average contraction direction. If `directionRenderType = INTEGRATION_POINT`, a separate direction line is drawn per point.
Table 6.6: FEM-specific rendering properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>elementWidgetSize</td>
<td>size of element to render ( \in [0, 1] )</td>
</tr>
<tr>
<td>directionRenderLen</td>
<td>relative length to draw fibre direction indicator ( \in [0, 1] )</td>
</tr>
<tr>
<td>directionRenderType</td>
<td>where to draw directions: ELEMENT, INTEGRATION_POINT</td>
</tr>
<tr>
<td>surfaceRendering</td>
<td>how to render surface: None, Shaded, Stress, Strain, MAPStress, MAPStrain, MaxShearStress, MaxShearStrain</td>
</tr>
<tr>
<td>stressPlotRange</td>
<td>range of values for stress/strain plot</td>
</tr>
<tr>
<td>stressPlotRanging</td>
<td>how to determine stress/strain plot range: Auto, Fixed</td>
</tr>
<tr>
<td>colorMap</td>
<td>delegate object controlling the map of stress/strain values to color</td>
</tr>
</tbody>
</table>

The last four properties apply to FemModel3d and FemMeshComp. They control how the surface is colored. This can be used to enable stress/strain visualizations. The property `surfaceRendering` sets what to draw:

- None: no surface
- Shaded: the face color specified by the mesh’s RenderProps
- Stress: the von Mises stress
- Strain: the von Mises strain
- MAPStress: Maximum absolute value principal stress
- MAPStrain: Maximum absolute value principal strain
- MaxShearStress: Maximum shear stress
- MaxStearStrain: Maximum sheer strain

The `stressPlotRange` controls the range of values to use when plotting stress/strain. Values outside this range are truncated. The `colorMap` is a delegate object that converts those stress and strain values to colors. Various types of maps exist, including GreyscaleColorMap, HueColorMap, RainbowColorMap, and JetColorMap. These all implement the ColorMap interface.

To display values corresponding to colors, a ColorBar needs to be added to the RootModel. Color bars are general Renderable objects that are only used for visualizations. They are added to the display using the `addRenderable` method in RootModel. Color bars also have a ColorMap associated with it. The following functions are useful for controlling its visualization:

```java
setNumberFormat ( String fmtStr );    // C-like numeric format specification
populateLabels ( double min, double max, int tick ); // initialize labels
updateLabels ( double min, double max ); // update existing labels
setColorMap ( ColorMap map );        // set color map

// Control position/size of the bar
setNormalizedLocation (double x, double y, double width, double height);
setLocationOverride (double x, double y, double width, double height);
```

The normalized location specifies sizes relative to the screen size (1 = screen width/height). The location override, if values are non-zero, will override the normalized location, specifying values in absolute pixels. Negative values for position correspond to distances from the left/top. For instance,

```java
setNormalizedLocation (0, 0.1, 0, 0.8); // set relative positions
setLocationOverride (-40, 0, 20, 0);   // override with pixel lengths
```

will create a bar that is 10% up from the bottom of the screen, 40 pixels from the right edge, with a height occupying 80% of the screen, and width 20 pixels.

Note that the color bar is not associated with any mesh or finite element model. Any synchronization of colors and labels must be done manually by the developer. It is recommended to do this in the RootModel’s `prerender(...)` method, so that colors are updated every time the model’s rendering configuration changes.
6.11.1 Example: stress and strain plotting

Figure 6.17: FemBeamColored model loaded into ArtiSynth.

The following model extends FemBeam to render stress, with an added color bar. The loaded model is shown in Figure 6.17.

```java
package artisynth.demos.tutorial;

import java.io.IOException;
import maspack.render.RenderList;
import maspack.util.DoubleInterval;
import artisynth.core.femmodels.FemModel.Ranging;
import artisynth.core.femmodels.FemModel.SurfaceRender;
import artisynth.core.renderables.ColorBar;

public class FemBeamColored extends FemBeam {

    @Override
    public void build(String[] args) throws IOException {
        super.build(args);

        // Show stress on the surface
        fem.setSurfaceRendering(SurfaceRender.Stress);
        fem.setStressPlotRanging(Ranging.Auto);

        // Create a colorbar
        ColorBar cbar = new ColorBar();
        cbar.setName("colorBar");
        cbar.setNumberFormat("%.2f");  // 2 decimal places
        cbar.populateLabels(0.0, 1.0, 10);  // Start with range [0,1], 10 ticks
        cbar.setLocation(-100, 0.1, 20, 0.8);
        addRenderable(cbar);

    }

    @Override
    public void prerender(RenderList list) {
        super.prerender(list);
        // Synchronize color bar/values in case they are changed. Do this *after*
        // super.prerender(), in case values are changed there.
        ColorBar cbar = (ColorBar)(renderables().get("colorBar"));
        cbar.setColorMap(fem.getColorMap());
        DoubleInterval range = fem.getStressPlotRange();
        cbar.updateLabels(range.getLowerBound(), range.getUpperBound());
    }
}
```
40
41
42  }
43
44  }
Chapter 7

Contact and Collision

ArtiSynth supports contact and collisions between various collidable bodies, including rigid bodies and FEM models, through a collision handling mechanism built into MechModel. Collisions are disabled by default, but can be enabled between specific pairs of bodies, or between general categories of rigid and deformable bodies (Section 7.1). Collisions are supported between any body that implements the interface Collidable.

Collision detection works by finding the intersecting regions between the collision meshes of collidable bodies, and then using the intersection information to determine contact points and their associated constraints (Section 7.4). Collision meshes must be instances of PolygonalMesh and must be triangular; mesh requirements are described in more detail in Section 7.3. Typically, a collision mesh is the same as the body’s surface mesh, but other meshes (or collections of meshes) can be specified instead (Section 7.3).

Detailed aspects of the collision behavior, such as friction, optional compliance and damping terms, methods used to determine contacts, and various tolerances, can be controlled using CollisionBehavior objects (Section 7.2). Collisions can be visualized by rendering contact normals, forces, intersection contours, and contact pressures (Section 7.5), and information about specific collisions, including contact data, forces, and pressures, can also be monitored as the simulation proceeds (Section 7.8).

It is also possible to explicitly control contact forces by making them an explicit function of penetration depth (Section 7.7); in particular, this capability is used to support elastic foundation contact (Section 7.7.3).

It should be understood that collision handling can be computationally expensive and, due to its discontinuous nature, may be less accurate than other aspects of the simulation. ArtiSynth therefore provides a number of ways to selectively control collision handling between different pairs of bodies. Collision handling is also challenging because if collisions are enabled among $n$ objects, then one needs to be able to easily specify the characteristics of up to $O(n^2)$ collision interactions, while also managing the fact that these interactions are highly transient. In ArtiSynth, collision handling is managed by a CollisionManager that is a subcomponent of each MechModel. The collision manager maintains default collision behaviors among certain groups of collidable objects, while also allowing the user to override the default behaviors by setting specific behaviors for any given pair of collidables.

Within ArtiSynth, the terms collision and contact are used somewhat interchangeably, although we acknowledge that in the literature, collision is generally understood to refer to the transient process that occurs when bodies first come into contact, while contact refers to the more persistent situation as the bodies remain together.

7.1 Enabling collisions

This section describes how to enable collisions in code. However, it is also possible to set some aspects of collision behavior using the ArtiSynth GUI. See the section “Collision handling” in the ArtiSynth User Interface Guide.

ArtiSynth can simulate collisions between bodies that implement the interface Collidable. Collidable bodies are further subdivided into those that are rigid and those that are deformable, according to whether their isDeformable() method returns true. Rigid collidables include RigidBody, while deformable collidables include FEM models (FemModel3d),
their mesh components (FemMeshComp), and skinned meshes (SkinMeshBody, Chapter 9). Because of the computational cost, collision detection is turned off by default and must be explicitly enabled when a model is built. Collisions can be enabled between specific pairs of bodies, or between general groupings of rigid and deformable bodies.

Collision handling is implemented by a model’s MechModel, which contains a CollisionManager subcomponent that keeps track of which bodies are colliding and computes the constraints and forces needed to enforce the collision behaviors. The collision manager itself can be accessed by the MechModel method

```java
getCollisionManager()
```

and can be used to query and set various collision properties, as described further below.

### 7.1.1 Collisions between specific bodies

As indicated above, collidable bodies are typically components such as rigid bodies or FEM models. Given two collidable bodies `collidable0` and `collidable1`, the simplest way to enable collisions between them is with the MechModel methods

```java
setCollisionBehavior (collidable0, collidable1, enabled)
setCollisionBehavior (collidable0, collidable1, enabled, mu)
```

The first method enables collisions without friction, while the second enables collisions with friction as specified by `mu`, which is the coefficient of Coulomb (or dry) friction. The `mu` value is ignored if `enabled` is false. Specifying a `mu` value of 0 disables friction, and the friction coefficient can also be left undefined by specifying a `mu` value less than 0, in which case the coefficient is inherited from the global friction value accessed by the MechModel methods

```java
setFriction (mu)
double getFriction ()
```

More detailed control over collision behavior can be achieved using the method

```java
setCollisionBehavior (collidable0, collidable1, behavior)
```

where `behavior` is a CollisionBehavior object (Section 7.2.1) that specifies both `enabled` and `mu`, along with other, more detailed collision properties (see Section 7.2.1).

The `setCollisionBehavior()` methods work by adding a CollisionBehavior object to the collision manager as a subcomponent. With the method `setCollisionBehavior(collidable0,collidable1,behavior)`, the behavior object is created and supplied by the application. With the other methods, the behavior object is created automatically and returned by the method. Once a behavior has been specified, it can then be queried using

```java
CollisionBehavior getCollisionBehavior (collidable0, collidable1)
```

This method will return `null` if no behavior for the pair in question has been explicitly set using one of the `setCollisionBehavior()` methods.

One should normally avoid enabling collisions between bodies that are otherwise connected, for example, adjacent bodies in a linkage connected by joints, in order to avoid conflicts between the connection and the collision behavior. If collision interaction is required between parts of two connected bodies, this can be achieved in various ways as described in Section 7.3.

Because behaviors are proper components, it is not permissible to add them to the collision manager twice. Specifically, the following will produce an error:

```java
CollisionBehavior behav = new CollisionBehavior();
behav.setDrawIntersectionContours (true);
mech.setCollisionBehavior (col0, col1, behav);
mech.setCollisionBehavior (col2, col3, behav); // ERROR
```

However, if desired, a new behavior can be created from an existing one:
CollisionBehavior behav = new CollisionBehavior();
behav.setDrawIntersectionContours (true);
mech.setCollisionBehavior (col0, col1, behav);
behav = new CollisionBehavior(behav);
mech.setCollisionBehavior (col2, col3, behav); // OK

7.1.2 Default collisions between groups

For convenience, it is also possible to specify default collision behaviors between different groups of collidables.

The default collision behavior between all collidables can be controlled using the MechModel methods

```java
mech.setDefaultCollisionBehavior (enabled, mu);
mech.setDefaultCollisionBehavior (behavior);
```

where enabled, mu and behavior act as described in Section 7.1.1.

Because of the possible computational expense of collision detection, default collision behaviors should be used with care.

In addition, a default collision behavior can be set for generic groups of collidables using the MechModel methods

```java
mech.setDefaultCollisionBehavior (group0, group1, enabled, mu);
mech.setDefaultCollisionBehavior (group0, group1, behavior);
```

where group0 and group1 are static instances of Collidable.Group that represent the groups described in Table 7.1. The groups Collidable.Rigid and Collidable.Deformable denote collidables that are rigid and deformable, respectively. The group Collidable.AllBodies denotes both rigid and deformable bodies, and Collidable.Self is used to enable self-collision, which is described in greater detail in Section 7.3.2.

<table>
<thead>
<tr>
<th>Collidable group</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collidable.Rigid</td>
<td>rigid collidables (e.g., rigid bodies)</td>
</tr>
<tr>
<td>Collidable.Deformable</td>
<td>deformable collidables (e.g., FEM models)</td>
</tr>
<tr>
<td>Collidable.AllBodies</td>
<td>rigid and deformable collidables</td>
</tr>
<tr>
<td>Collidable.Self</td>
<td>enables self-intersection for compound deformable collidables</td>
</tr>
<tr>
<td>Collidable.All</td>
<td>rigid and deformable collidables and self-intersection</td>
</tr>
</tbody>
</table>

Table 7.1: Collision group types.

A call to one of the mech.setDefaultCollisionBehavior() methods will override the effects of previous calls. So for instance, the code sequence

```java
mech.setDefaultCollisionBehavior (true, 0);
mech.setDefaultCollisionBehavior (Collidable.Deformable, Collidable.Rigid, false, 0);
mech.setDefaultCollisionBehavior (true, 0.2);
```

will initially enable collisions between all bodies with a friction coefficient of 0, then disable collisions between deformable and rigid bodies, and finally re-enable collisions between all bodies with a friction coefficient of 0.2.

The default collision behavior between any pair of collidable groups can be queried using

```java
CollisionBehavior getDefaultValueBehavior (group0, group1)
```

where group0 and group1 are restricted to the primary groups Collidable.Rigid, Collidable.Deformable, and Collidable.Self, since individual behaviors are not maintained for the composite groups Collidable.AllBodies and Collidable.All.

Specific collision behaviors set using the mech.setCollisionBehavior() methods of Section 7.1.1 will override any default collision settings. In addition, the second argument collidables1 of these methods can describe either an individual collidable, or one of the groups of Table 7.1. For example, the code fragment
MechModel mech;
RigidBody bodA;
FemModel3d femB;
...

mech.setCollisionBehavior (bodA, Collidable.Deformable, true, 0.1);
mech.setCollisionBehavior (femB, Collidable.AllBodies, true, 0.0);
mech.setCollisionBehavior (bodA, femB, false, 0.0);

will enable collisions between bodA and all deformable collidables (with friction 0.1), as well as femB and all deformable and rigid collidables (with friction 0.0), while specifically disabling collisions between bodA and femB.

To determine the actual behavior controlling collisions between two collidables (whether due to a default behavior or one specified using setCollisionBehavior()), one may use the method

getActingCollisionBehavior (collidable0, collidable1)

where collidable0 and collidable1 must both be specific collidable components and cannot be a group (such as Collidable.Rigid or Collidable.All). If one of the collidables is a compound collidable (Section 7.3.2), or has a collidability setting (Section 7.2.2) that prevents collisions, there may be no consistent acting behavior, in which case the method returns null.

Collision behaviors take priority over each other in the following order:


2. Behaviors specified using setCollisionBehavior() involving one specific collidable and a group of collidables (indicated by a Collidable.Group), with later specifications taking priority over earlier ones.

3. Default behaviors specified using setDefaultCollisionBehavior().

A collision behavior specified with setCollisionBehavior() can later be removed using

clearCollisionBehavior (collidable0, collidable1)

and all such behaviors in a MechModel can be removed with

clearCollisionBehaviors ()

Note that this latter call does not remove default behaviors specified with setDefaultCollisionBehavior().

7.1.3 Example: collision with a plane

A simple model illustrating collision between two jointed rigid bodies and a plane is defined in

artisynth.demos.tutorial.JointedCollide

This model is simply a subclass of RigidBodyJoint that overrides the build() method to add an inclined plane and enable collisions between it and the two connected bodies:

```java
public void build (String[] args) {
    super.build (args);
    bodyB.setDynamic (true); // allow bodyB to fall freely
    // create and add the inclined plane
    RigidBody base = RigidBody.createBox ("base", 25, 25, 2, 0.2);
    base.setPose (new RigidTransform3d (5, 0, 0, 1, 0, -Math.PI/8));
    base.setDynamic (false);
    mech.addRigidBody (base);
}
// turn on collisions
mech.setDefaultCollisionBehavior (true, 0.20);  
mech.setCollisionBehavior (bodyA, bodyB, false);
}

The superclass build() method called at line 3 creates everything contained in RigidBodyJoint. The remaining code then alters that model: bodyB is set to be dynamic (line 5) so that it will fall freely, and an inclined plane is created from a thin box that is translated and rotated and then set to be non-dynamic (lines 8-11). Finally, collisions are enabled by setting the default collision behavior (line 14), and then specifically disabling collisions between bodyA and bodyB (line 15). As indicated above, the latter step is necessary because the joint would otherwise keep the two bodies in a permanent state of collision.

To run this example in ArtiSynth, select All demos > tutorial > JointedCollide from the Models menu. The model should load and initially appear as in Figure 7.1. Running the model (Section 1.5.3) will cause the jointed assembly to collide with and slide off the inclined plane.

![JointedCollide model loaded into ArtiSynth.](image)

Figure 7.1: JointedCollide model loaded into ArtiSynth.

### 7.1.4 Collisions for FEM models

Both FemModel3d and FemMeshComp implement Collidable. By default, a FemModel3d uses its surface mesh as the collision surface, while FemMeshComp will uses the mesh it contains (although collisions will only occur if this is a triangular polygonal mesh).

Because FemModel3d contains other collidables as subcomponents, it is considered a compound collidable, as discussed further in Section 7.3.2. In particular, since FemMeshComp is also a Collidable, we can enable collisions with any embedded mesh inside an FEM. Any forces resulting from the collision are then automatically transferred back to the underlying nodes of the model using Equation (6.5).

**Note:** Collisions involving shell elements are not yet fully supported. This relates to the fact that shells are thin and can therefore pass through each other easily in a single time step, and also that meshes associated with shell elements are usually not closed. However, collisions should work properly if

1. The collision meshes do not pass completely through each other in a single time step;
2. Collisions do not occur near open mesh edges.

Restriction 2 can often be relaxed if the collider type is set to TRI_INTERSECTION (Section 7.4.2).
7.1.5 Example: FEM models and rigid bodies

![Figure 7.2: FemCollisions model loaded into ArtiSynth.](image)

An example of FEM collisions is shown in Figure 7.2. The full source code can be found in the ArtiSynth repository under artisynth.demos.tutorial.FemCollisions. The model contains a rigid body table and three FEM models: a beam (blue), an ellipsoid (red), and a hex block containing an embedded spherical mesh (green). The collision-enabling code is as follows:

```csharp
// Set up collisions
mech.setCollisionBehavior(ellipsoid, beam, true); // beam-ellipsoid
mech.setCollisionBehavior(ellipsoid, table, true); // ellipsoid-table
mech.setCollisionBehavior(table, beam, true); // beam-table

FemMeshComp embeddedSphere = block.getMeshComp("embedded"); // get embedded FemMeshComp
mech.setCollisionBehavior(embeddedSphere, table, true); // sphere-table
mech.setCollisionBehavior(ellipsoid, embeddedSphere, true); // sphere-ellipsoid
```

This enables collisions between the ellipsoid and the beam, table and embedded sphere, and between the table and the beam and embedded sphere. However, collisions are not enabled between the block itself and any other components; notice in the figure that the block surface passed through the table and ellipsoid.

7.2 Collision behaviors and collidability

7.2.1 Collision behaviors

As mentioned above, `CollisionBehavior` objects can be used to control other aspects of contact beyond friction and enabling. This may be done by setting the `CollisionBehavior` properties listed below. Except where otherwise indicated, these properties are also exported by the collision manager, where they can be used to provide global default settings for all collisions.

In addition to these properties, `CollisionBehavior` and `CollisionManager` export other properties that can be used to control the rendering of collisions, as described in Section 7.5.

enabled

A boolean that determines if collisions are enabled. Not present in the collision manager.

friction
A double giving the coefficient of Coulomb friction, typically in the range \([0, 0.5]\). The default value is 0. Setting friction to a non-zero value increases the simulation time, since extra constraints must be added to the system to accommodate the friction.

**method**

An instance of `CollisionBehavior.Method` that controls how the contact constraints for the collision response are generated. The methods, described in more detail in Section 7.4.1, are:

<table>
<thead>
<tr>
<th>Method</th>
<th>Constraint generation</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONTOUR_REGION</td>
<td>constraints generated from planes fit to each mesh contact region</td>
</tr>
<tr>
<td>VERTEX_PENETRATION</td>
<td>constraints generated from penetrating vertices</td>
</tr>
<tr>
<td>VERTEX_EDGE_PENETRATION</td>
<td>constraints generated from penetrating vertices and edge-edge contacts</td>
</tr>
<tr>
<td>DEFAULT</td>
<td>method is determined automatically</td>
</tr>
<tr>
<td>INACTIVE</td>
<td>no constraints generated</td>
</tr>
</tbody>
</table>

The default value is `DEFAULT`.

**vertexPenetrations**

An instance of `CollisionBehavior.VertexPenetrations` that controls the collidables for which vertex penetrations are computed when using vertex penetration contact (see Sections 7.4.1.2 and 7.4.1.3). The default value is `AUTO`.

**bilateralVertexContact**

A boolean which causes the system to handle vertex penetration contact using bilateral constraints instead of unilateral constraints. This usually improves computational performance significantly for collisions involving FEM models, but may result in overconstrained contact when vertex penetration contact is used between rigid bodies, as well as in some other circumstances (Section 7.6). The default value is `true`.

**colliderType**

An instance of `CollisionManager.ColliderType` that specifies the underlying mechanism used to determine the collision information between two meshes. The choice of collider may restrict which collision methods (described above) are allowed. Collider types are described in more detail in Section 7.4.1 and include:

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AJL_CONTOUR</td>
<td>uses mesh intersection contour to find penetrating regions and vertices</td>
</tr>
<tr>
<td>TRI_INTERSECTION</td>
<td>uses triangle intersections to find penetrating regions and vertices</td>
</tr>
<tr>
<td>SIGNED_DISTANCE</td>
<td>uses a signed distance field to find penetrating vertices</td>
</tr>
</tbody>
</table>

The default value is `TRI_INTERSECTION`.

**reduceConstraints**

A boolean which, if `true`, indicates that the system should try to reduce the number of contacts between bodies in order to try and remove redundant contacts. See Section 7.6. The default value is `false`.

**compliance**

A double which adds a compliance (inverse stiffness) to the collision behavior, so that the contact has a “springiness”. See Section 7.6. The default value for this is 0 (no compliance).

**damping**

A double which, if `compliance` is non-zero, specifies a damping to accompany the compliant behavior. See Section 7.6. The default value is 0. When compliance is specified, it is usually necessary to set the damping to a non-zero value to prevent bouncing.

**frictionCreep**

A double which, if non-zero, is used to regularize friction constraints. The default value is 0. It is usually only necessary to set this number when `MechModel`'s `useImplicitFriction` property is set to `true` (see Section 7.9.4), and when the contact constraints are redundant (Section 7.6). The number should be interpreted as a small “creep” speed with which contacts nominally held still by static friction are allowed to drift. The number should be as large as possible without seriously affecting the simulation.

**forceBehavior**

A composite property of type `ContactForceBehavior` specifying a contact force behavior, as described in Section 7.7.2. The default value is `null`.
penetrationTol

A double controlling the amount of interpenetration that is permitted in order to ensure contact stability (see Section 7.9). This property is not present in the collision manager. If unspecified, the system will inherit this property from the MechModel, which computes a default penetration tolerance based on the overall size of the model.

rigidRegionTol

A double which, for the CONTOUR_REGION contact method and the TRI_INTERSECTION collider type, specifies an overlap factor that is used to group individual triangle intersections into contact regions. If not explicitly specified, the system computes a default value based on the overall size of the model.

rigidPointTol

A double which, for the CONTOUR_REGION contact method, specifies a minimum distance between contact points that is used to reduce the number of contacts. If not explicitly specified, the system computes a default value for this based on the overall size of the model.

To set properties globally in the collision manager, one can access it using the MechModel method `getCollisionManager()`, and then employ a code fragment such as the following:

```java
CollisionManager cm = mech.getCollisionManager();
cm.setReduceConstraints (true);
```

Since collision behaviors are subcomponents of the collision manager, properties set in the collision manager will be *inherited* by any behaviors for which they have not been explicitly set.

One can also set properties using the GUI, by selecting the collision manager or a collision behavior in the navigation panel and then selecting *Edit properties ...* from the context menu. See “Property panels” and “Collision handling” in the ArtiSynth User Interface Guide.

To set properties for specific collidable pairs, one can call either `setDefaultCollisionBehavior()` or `setCollisionBehavior()` with an appropriately set `CollisionBehavior` object:

```java
RigidBody bodA;
CollisionBehavior behav = new CollisionBehavior (enabled, mu);
beHAV.setPenetrationTol (0.001);
setDefaultCollisionBehavior (Collidable.Deformable, Collidable.Rigid, behav);

beHAV.setPenetrationTol (0.003);
setCollisionBehavior (bodA, Collidable.Rigid, behav);
```

For behaviors that are already set, one may use `getDefaultCollisionBehavior()` or `getCollisionBehavior()` to obtain the behavior and then set the desired properties directly:

```java
RigidBody bodA;
CollisionBehavior behav;

beHAV = getDefaultCollisionBehavior (Collidable.Deformable, Collidable.Rigid);
beHAV.setPenetrationTol (0.001);
beHAV = getCollisionBehavior (bodA, Collidable.Rigid);
beHAV.setPenetrationTol (0.003);
```

Note however that `getDefaultCollisionBehavior()` only works for `Collidable.Rigid, Collidable.Deformable, and Collidable.Self`, and that `getCollisionBehavior()` only works for a collidable pair that has been previously specified with `setCollisionBehavior()`. One may also use `getActingCollisionBehavior()` (described above) to obtain the behavior (default or otherwise) responsible for a specific pair of collidables, although in some instances no such single behavior exists and the method will then return `null`.

There are two constructors for `CollisionBehavior`:

```java
CollisionBehavior ()
CollisionBehavior (boolean enable, double mu)
```
The first creates a behavior with the enabled property set to false and other properties set to their default (generally inherited) values. The second creates a behavior with the enabled and friction properties explicitly specified, and other properties set to their defaults. If \( \mu \) is less than zero or enabled is false, then the friction property is set to be inherited so that its value is controlled by the global friction property in the collision manager (and accessed using the MechModel methods setFriction(\( \mu \)) and getFriction()).

### 7.2.2 Collidability

Each collidable component maintains a collidable property, which specifically enables or disables the ability of that collidable to collide with other collidables.

The collidable property value is of the enumerated type Collidable.Collidability and has four possible settings:

- **OFF**
  
  All collisions disabled: the collidable will not collide with anything.

- **ALL**
  
  All collisions (both internal and external) enabled: the collidable may collide with any other collidable.

- **INTERNAL**
  
  The collidable may only collide with other collidables that are inside the same collidable hierarchy of some compound collidable (Section 7.3.2).

- **EXTERNAL**
  
  The collidable may collide with any other collidable except those that are inside the same collidable hierarchy of some compound collidable.

Note that collidability only enables collisions. In order for collisions to actually occur between two collidables, a default or override collision behavior must also be specified for them in the MechModel.

### 7.3 Collision meshes and compound collidables

Contact between collidable bodies is determined by finding intersections between their collision meshes. Collision meshes must be instances of PolygonalMesh, and must also be triangular, manifold, oriented, and non-self-intersecting (at least in the region of contact). The oriented requirement helps the collision detector differentiate inside from outside. Collision meshes should also be closed, if possible, although collision may sometimes work with the open meshes (such as those that arise with shell elements) under conditions described in Section 7.1.4. Collision meshes do not need to be connected; a collision mesh may be composed of separate parts.

Commonly, a collidable body has a single collision mesh which is the same as its surface mesh. However, some collidables, such as rigid bodies and FEM models, allow an application to specify a collidable mesh that is different from its surface mesh. This can be useful in situations such as the following:

- Collisions should be enabled for only part of a collidable, perhaps because other parts are in contact due to joints or other types of attachment.

- The collision mesh requires a different resolution than the surface mesh, possibly for computational reasons.

- The collision mesh requires a different geometry, such as in situations where the collidable’s physical surface is sufficiently thin that it may otherwise pass through other collidables undetected (Section 7.9.2).

For a rigid body, the collision mesh is returned by its getCollisionMesh() method (see Section 7.3.4). By default, this is the same as the surface mesh returned by getSurfaceMesh(). However, the collision mesh can be changed by adding one (or more) additional mesh components to the meshes component list and setting its isCollidable property to true, usually while also setting isCollidable to false for the surface mesh (Section 3.2.9). The collision mesh returned by getCollisionMesh() is then formed from the union of all rigid body meshes for which isCollidable is true.
The *sum* operation used to create the `RigidBody` collision mesh uses `addMesh()`, which simply adds all vertices and faces together. While the result works correctly for collisions, it does not represent a proper CSG union operation (such as that described in Section 2.5.7) and may contain interpenetrating and overlapping features. Care should be taken to ensure that the resulting mesh is not self-intersecting in the regions that will be exposed to contact.

For FEM models, the mechanism is a little different, as discussed below in Section 7.3.2. An FEM model can have *multiple* collision meshes, depending on the setting of the collidable property of each of its mesh components. The ability to have multiple collision meshes permits self-collision intersection of the FEM model with itself.

### 7.3.1 Example: redefining a rigid body collision mesh

A model illustrating how to redefine the collision mesh for a rigid body is defined in `artisynth.demos.tutorial.JointedBallCollide` like `JointedCollide` in Section 7.1.3, this model is simply a subclass of `RigidBodyJoint` that overrides the `build()` method, adding a ball to the tip of `bodyA` to enable it to collide with `bodyB`:

```java
public void build (String[] args) {
    super.build (args); // build the RigidBodyJoint model
    // create a ball mesh
    PolygonalMesh ball = MeshFactory.createIcosahedralSphere (2.5, 1);
    // translate it to the tip of bodyA, add it to bodyA, and make it blue-gray
    ball.transform (new RigidTransform3d (5, 0, 0));
    RigidMeshComp mcomp = bodyA.addMesh (ball);
    RenderProps.setFaceColor (mcomp, new Color (.8f, .8f, 1f));
    // disable collisions for the main surface mesh of bodyA
    bodyA.getSurfaceMeshComp().setIsCollidable (false);
    // enable collisions between bodyA and bodyB
    mech.setCollisionBehavior (bodyA, bodyB, true);
}
```

The superclass `build()` method called at line 3 creates everything contained in `RigidBodyJoint`. The remaining code then alters that model: A ball mesh is created, translated to the tip of `bodyA`, added as an additional mesh (Section
3.2.9), and set to render as blue-gray (lines 5-10). Next, collisions are disabled for bodyA’s main surface mesh by setting its isCollidable property to false (line 13); this will ensure that the collision mesh associated with bodyA will consist solely of the ball mesh, which is necessary because the surface mesh is permanently in contact with bodyB. Lastly, collisions are enabled between bodyA and bodyB (line 15).

To run this example in ArtiSynth, select All demos > tutorial > JointedBallCollide from the Models menu. Running the model will cause bodyA to fall, pivot about the hinge joint, and collide with bodyB (Figure 7.3).

7.3.2 Compound collidables and self-collision

An FEM model is an example of a compound collidable, which may contain subcollidable descendant components which are also collidable. Compound collidables are identified by having their isCompound() method return true. For an FEM model, the subcollidables are the mesh components in its meshes list. A non-compound collidable which is not a subcollidable of some other (compound) collidable is called solitary. If A is a subcollidable of a compound collidable C, then C is an ancestor collidable of A.

Subcollidables do not need to be immediate child components of the compound collidable; they need only be descendants.

One of the main purposes of compound collidables is to enable self-collisions. While ArtiSynth does not currently support the detection of self-collisions within a single mesh, self-collision can be implemented within a compound collidable C by detecting collisions amount the (separate) meshes of its subcollidables.

Compound collidables and their subcollidables are assumed to be deformable; otherwise, subcollision would not be possible.

Figure 7.4: A collection of collidable components, where A is compound, with subcollidables A1, A2, and A3, B is solitary, and C is compound with subcollidables C1 and C2. The non-compound collidables, including the leaf nodes of the collidable trees, are also instances of CollidableBody, indicated by a double outline.

In general, an ArtiSynth model will contain a number of collidable components, some of which are compound (Figure 7.4). The non-compound collidables, including both solitary collidables and leaf nodes of a compound collidable’s tree, are also instances of the subinterface CollidableBody, which provide the collision meshes, along with other information used to compute the collision response (Section 7.3.4).

Actual collisions happen only between CollidableBodys; compound collidables are used only for determining grouping and specifying collision behaviors. The rules for determining whether two collidable bodies A and B will actually collide are as follows:

1. Internal (self) collisions: If A and B are both subcollidables of some compound collidable C, then A and B will collide if (a) both their collidable properties are set to either ALL or INTERNAL, and (b) an explicit collision behavior is set between A and B, or self-collision is enabled for C (as described below).

2. External collisions: Otherwise, if A and B are either solitary or subcollidables of different compound collidables, then they will collide if (a) both their collidable properties are set to either ALL or EXTERNAL, and (b) a specific or default collision behavior is set between them or their collidable ancestors.
As mentioned above, the subcollidables of an FEM are its mesh components (Section 6.3), each of which is a collidable implemented by `FemMeshComp`. When a mech component is added to an FEM model, using either `addMeshComp()` or one of the `addMesh()` methods, its collidable property is automatically set to `INTERNAL`, so if a different setting is required, this must be specified after the component has been added to the model.

Subject to the above conditions, self-collision can be enabled for a specific compound collidable C by calling the `setCollisionBehavior()` methods with `collidable0` set to C and `collidable1` set to either C or `Collidable.SELF`, as in for example:

```java
mech.setCollisionBehavior(C, Collidable.SELF, true, mu);
... OR ...
mech.setCollisionBehavior(C, C, true, mu);
```

It can also be enabled, by default, for all compound collidables by calling one of the `setDefaultCollisionBehavior()` methods with `collidable0` set to `Collidable.DEFORMABLE` and `collidable1` set to `Collidable.SELF`, e.g.:

```java
mech.setDefaultCollisionBehavior(Collidable.DEFORMABLE, Collidable.SELF, true, mu);
```

For an example of how collision interactions can be set, refer to Figure 7.4. Assume that components A, B and C are deformable, and that the collidable property for all collidables is set to `ALL` except for A3, where it is set to `EXTERNAL` (implying that A3 cannot self-collide with A1 and A2). Then if `mech` is the `MechModel` containing the collidables, the call

```java
mech.setDefaultCollisionBehavior(Collidable.DEFORMABLE, Collidable.DEFORMABLE, true, 0.2);
```

will enable collisions between A1, A2, and A3 and each of B, C1, and C2, and between B and C1 and C2, but not among A1, A2, and A3 or C1 and C2. The subsequent calls

```java
mech.setDefaultCollisionBehavior(Collidable.DEFORMABLE, Collidable.SELF, true, 0);
mech.setCollisionBehavior(B, A3, false);
```

will enable self-collision between A1 and A2 and C1 and C2 with zero friction, and disable collision between B and A3. Finally, the calls

```java
mech.setCollisionBehavior(A3, C, true, 0.3);
mech.setCollisionBehavior(A, A, false);
```

will enable collision between A3 and each of C1 and C2 with friction 0.3, and disable all self-collisions among A1, A2 and A3.

### 7.3.3 Example: FEM model with self-collision

A model illustrating self-collision for an FEM model is defined in

```java
artisynth.demos.tutorial.FemSelfCollide
```

It creates a simple FEM based on a partial torus that self intersects when it falls under gravity. Internal surface meshes are added to the left and right ends of the model to prevent interpenetration. The code for the `build()` method is listed below:

```java
1 public class FemSelfCollide extends RootModel {
2 3   public void build (String[] args) {
4     MechModel mech = new MechModel ("mech");
5     addModel (mech);
```
Figure 7.5: FemSelfCollide showing collision between the left and right contact meshes (left). Without these meshes, the FEM model intersects with itself as shown on the right.

```java
// create FEM model based on a partial (open) torus, with an
// opening (gap) of angle of PI/4.
FemModel3d ptorus = new FemModel3d("ptorus");
FemFactory.createPartialHexTorus(
    ptorus, 0.15, 0.03, 0.06, 10, 20, 2, 7*Math.PI/4);
// rotate the model so that the gap is at the bottom
ptorus.transformGeometry(
    new RigidTransform3d (0, 0, 0, 0, 3*Math.PI/8, 0));
// set material and particle damping
ptorus.setMaterial (new LinearMaterial (5e4, 0.45));
ptorus.setParticleDamping (1.0);
mech.addModel (ptorus);

// anchor the FEM by fixing the top center nodes
for (FemNode3d n : ptorus.getNodes()) {
    if (Math.abs(n.getPosition().x) < 1e-15) {
        n.setDynamic(false);
    }
}

// Create sub-meshes to resist collision at the left and right ends of the
// open torus. At each end, create a mesh component, and use its
// createVolumetricSurface() method to create the mesh from the
// elements near the end.
LinkedHashSet<FemElement3d> elems =
    new LinkedHashSet<>(); // elements for mesh building
FemMeshComp leftMesh = new FemMeshComp (ptorus, "leftMesh");
// elements near the left end have numbers in the range 180 - 199
for (int n=180; n<200; n++) {
    elems.add (ptorus.getElementByNumber (n));
} leftMesh.createVolumetricSurface (elems);
ptorus.addMeshComp (leftMesh);

FemMeshComp rightMesh = new FemMeshComp (ptorus, "rightMesh");
elems.clear();
// elements at the right end have numbers in the range 0 - 19
for (int n=0; n<20; n++) {
    elems.add (ptorus.getElementByNumber (n));
} rightMesh.createVolumetricSurface (elems);
ptorus.addMeshComp (rightMesh);
```
// Create a collision behavior and use it to enable self collisions for
// the FEM model. Since the model has low resolution and sharp edges, use
// VERTEX_EDGE_PENETRATION, which requires the AJL_CONTOUR collider type.
CollisionBehavior behav = new CollisionBehavior (true, 0);
behav.setMethod (CollisionBehavior.Method.VERTEX_EDGE_PENETRATION);
behav.setColliderType (CollisionManager.ColliderType.AJL_CONTOUR);
mech.setCollisionBehavior (ptorus, ptorus, behav);

// render properties: render the torus using element widgets
ptorus.setElementWidgetSize (0.8);
RenderProps.setFaceColor (ptorus, new Color (0.4f, 1.0f, 0.6f));

// enable rendering of the left and right contact meshes
leftMesh.setSurfaceRendering (SurfaceRender.Shaded);
rightMesh.setSurfaceRendering (SurfaceRender.Shaded);

RenderProps.setFaceColor (leftMesh, new Color (0.78f, 0.78f, 1.0f));
RenderProps.setFaceColor (rightMesh, new Color (0.78f, 0.78f, 1.0f));

The model creates an FEM model based on an open torus, using a factory method in FemFactory, and rotates it so that
the gap is located at the bottom (lines 7-18). The torus is then anchored by fixing the nodes located at the top-center
(lines 20-25). Next, mesh components are created to enforce self-collision at the left and right gap end points (lines
27-47). First, a FemMeshComp is created (Section 6.3), and then its createVolumetricSurface() method is used
to create a local surface mesh wrapping the elements specified in elems. When selecting the elements, we use the
convenient fact that for this particular FEM model, the elements near the left and right ends have numbers in the ranges
180…199 and 0…19, respectively.

Once the submeshes have been added to the FEM model, we create a collision behavior and use it to enable self-
collisions (lines 49-55). An explicit behavior is created so that we can enable the VERTEX_EDGE_PENETRATION contact
method (Section 7.4.1), because the meshes are coarse and the additional edge-edge collisions will improve behavior;
this method also requires the AJL_CONTOUR collider type. While self-collision is enabled by calling

mech.setCollisionBehavior (ptorus, ptorus, behav);

it could also have been enabled by calling

mech.setCollisionBehavior (ptorus, Collidable Self, behav);

... OR ...

mech.setCollisionBehavior (leftMesh, rightMesh, behav);

Note that there is no need to set the collidable property of the collision meshes since it is set to INTERNAL by default
when they are added to the FEM model.

Render properties are set at lines 57-64, with the torus rendered as light green and the collision meshes as blue-gray. The
torus is drawn using its elements widgets instead of its surface mesh, to prevent the latter from obscuring the collision
meshes.

To run this example in ArtiSynth, select All demos > tutorial > FemSelfCollide from the Models menu. Running the
model will cause the FEM model to self-collide as shown in Figure 7.5.

7.3.4 Collidable bodies

As mentioned in Section 7.3.2, non-compound collidables, including both solitary collidables and leaf nodes of a
compound collidable’s tree, are also instances of CollidableBody, which provide the actual collision meshes and other
information used to compute the collision response. This is done using various methods, including:

• PolygonalMesh getCollisionMesh()  
  Returns the actual surface mesh to be used for computing collisions.
• boolean hasDistanceGrid()
  If this method returns true, the body also maintains a signed distance grid for the mesh, which can be used by the collider type SIGNED_DISTANCE (Section 7.4.1).

• DistanceGridComp getDistanceGridComp()
  If hasDistanceGrid() returns true, this method return the distance grid.

• double getMass()
  Returns the mass of this body (or a suitable estimate), for use in automatically computing certain collision parameters.

• int getCollidableIndex()
  Returns the index of this collidable body within the set of all CollidableBodys in the MechModel. The index is determined by the body’s depth-first ordering within the model component hierarchy. For components within the same component list, this ordering will be determined by the order in which the components are added in the model’s build() method.

7.3.5 Nested MechModels

It is possible in ArtiSynth for one MechModel to contain other nested MechModels within its component hierarchy. This raises the question of how collisions within the nested models are controlled. The general rule for this is the following:

The collision behavior for two collidables colA and colB is determined by whatever behavior (either default or override) is specified by the lowest MechModel in the component hierarchy that contains both colA and colB.

![Figure 7.6: A MechModel containing collidables B1 and B2, nested within another containing collidables A1 and A2.](image)

For example, consider Figure 7.6 where we have a MechModel (mechB) containing collidables B1 and B2, nested within another MechModel (mechA) containing collidables A1 and A2. Then consider the following code fragment:

```java
mechB.setDefaultCollisionBehavior(true, 0.2);
mechA.setCollisionBehavior(B1, Collidable.AllBodies, true, 0.0);
mechA.setCollisionBehavior(B1, A1, false);
mechA.setCollisionBehavior(B1, B2, false); // Error!
```

The first line enables default collisions within mechB (with $\mu = 0$), controlling the interaction between B1 and B2. However, collisions are still disabled within mechA, meaning A1 and A2 will not collide either with each other or with B1 or B2. The second line enables collisions between B1 and any other body within mechA (i.e., A1 and A2), while the third
line disables collisions between \( B_1 \) and \( A_1 \). Finally, the fourth line results in an error, since \( B_1 \) and \( B_2 \) are both contained within \( \text{mechB} \) and so their collision behavior cannot be controlled from \( \text{mechA} \).

While it is not legal to specify a specific behavior for collidables contained in a \( \text{MechModel} \) from a higher level \( \text{MechModel} \), it is legal to create collision response components for the same pair within both models. So the following code fragment would be allowed and would create response components in both \( \text{mechA} \) and \( \text{mechB} \):

```java
mechB.setCollisionResponse (B1, B2);
mechA.setCollisionResponse (B1, B2);
```

### 7.4 Implementation

This section describes the technical details of the different ways in which collision are implemented in ArtiSynth. Knowledge of these details can be useful for choosing collision behavior property settings, understanding elastic foundation contact (Section 7.7.3), and interpreting the information provided by collision responses (Section 7.8.1) and collision rendering (Section 7.5).

The collision mechanism works by using a \textit{collider} (described further below) to locate the \textit{contact regions} where the (triangular) collision meshes of different collidables intersect each other. Figure 7.7 shows four contact regions between two meshes representing human teeth. Information about each contact region is then used to generate \textit{contact constraints}, according to a \textit{contact method}, that prevent further collision and resolve any existing interpenetration.

![Figure 7.7: Collisions between collidables are found by locating contact regions (yellow contours) between their surface meshes.](image)

#### 7.4.1 Contact methods

Contact constraints are computed from mesh contact regions using one of two primary contact methods:

##### 7.4.1.1 Contour region

Contact constraints are generated for each contact region by first fitting a plane to the region, and then selecting contact points around its perimeter such that their 2D projection into the plane forms a convex hull (Figure 7.8). A contract constraint is assigned to each contact point, with a constraint direction parallel to the plane normal and a penetration...
distance given by the maximum interpenetration of the region. Note that the contact points do not necessarily lie on the plane, but they all have the same normal and penetration distance.

This is the default method used between rigid bodies, since it does not depend on mesh resolution, and rigid body contact behavior depends only on the convex hull of the contacting regions. Since the number of contact constraints may exceed the number of degrees of freedom (DOF) of the contacting bodies, the constraints are supplied to the physics solver as unilateral constraints (Section 1.2). Using contact region constraints when one of the collidables is an FEM model will result in an error.

### 7.4.1.2 Vertex penetration

Contacts are created for each mesh vertex that penetrates the other mesh, with a contact normal directed toward the nearest point on the opposing mesh (Figure 7.9). Contact constraints are generated for each contact, with the constraint direction parallel to the normal and the penetration distance equal to the distance to the opposing mesh.

This is the default contact method when one or both of the collidables is an FEM model. Also by default, if only one of the collidables is an FEM model, vertex penetrations are computed only for the FEM collidable; penetrations of the rigid body into the FEM are not considered. Otherwise, if both collidables are FEM models, then vertex penetrations are computed for both bodies, resulting in two-way contact (Figure 7.10).

Because FEM models tend to have a large number of DOFs, the default behavior is to supply vertex penetration constraints to the physics solver as bilateral constraints (Section 1.2), removing them only between simulation time steps when the contact disappears or a separating force is detected. This results in much faster simulation times, particularly for FEM models, because it does not require solving a linear complementarity problem. However, this also means that using vertex penetration between rigid bodies, or other bodies with a low number of degrees of freedom, will typically result in an overconstrained system that must be handled using one of the techniques described in Section 7.6).

### 7.4.1.3 Setting the contact method

Contact methods can be specified by the method property in either the collision manager or behavior (Section 7.2.1). The property’s value is an instance of CollisionBehavior.Method, for which the standard settings are:

**CONTOUR_REGION**

Contact constraints are generated using contour regions, as described in Section 7.4.1.1. Setting this for contact that involves one or more FEM models will result in an error.

**VERTEX_PENETRATION**

Contact constraints are generated using vertex penetration, as described in Section 7.4.1.2. Setting this for contact between rigid bodies may result in an overconstrained system.
**VERTEX_EDGE_PENETRATION**

An experimental version of vertex penetration contact that supplements vertex constraints with extra constraints formed from edge-edge contact. Intended for collision meshes with low mesh resolutions and sharp edges. Available only when using the `AJL_CONTOUR` collider type (Section 7.4.2).

**DEFAULT**

Selects `CONTOUR_REGION` when both collidables are rigid bodies and `VERTEX_PENETRATION` otherwise.

**INACTIVE**

No constraints are generated. This will result in no collision response.

![Figure 7.9: Vertex penetration contact creates a contact point for each penetrating vertex within a penetration region, with the contact normal directed toward the nearest point on the opposite mesh.](image1)

![Figure 7.10: Two-way vertex penetration contact involves creating contact points from the penetrating vertices of both meshes.](image2)

When vertex penetration contact is employed, the collidables for which penetrations are generated can also be controlled using the `vertexPenetrations` property, whose value is an instance of `CollisionBehavior.VertexPenetrations`:

**BOTH_COLLIDABLES**

Vertex penetrations are computed for both collidables.

**FIRST_COLLIDABLE**

Vertex penetrations are computed for the first collidable.

**SECOND_COLLIDABLE**

Vertex penetrations are computed for the second collidable.
Vertex penetrations are determined automatically as follows: for both collidables if neither is a rigid body, for the non-rigid collidable if one collidable is a rigid body, and for the first collidable otherwise.

## 7.4.2 Collider types

The contact regions used by the contact method are generated by a collider, which is specified by the `colliderType` property in either the collision manager (as the default), or in the collision behavior for a specific pair of collidables. The property’s value is an instance of `CollisionManager.ColliderType`, which describes the three collider types presently supported:

### TRI_INTERSECTION

The original ArtiSynth collider which uses a bounding-box hierarchy to locate all triangle intersections between the two surface meshes. Contact regions are then estimated by grouping the intersection points together, and penetrating vertices are computed separately using point-mesh distance queries based on the bounding-box hierarchy. This latter step requires iterating over all mesh vertices, which may be slow for large meshes. `TRI_INTERSECTION` can often handle intersections near the edges of an open mesh, but does not support the `VERTEX_EDGE_PENETRATION` contact method.

### AJL_CONTOUR

A bounding-box hierarchy is used to locate all triangle intersections between the two surface meshes, and the intersection points are then connected, using robust geometric predicates, to find the (piecewise linear) intersection contours. The contours are then used to identity the contact regions on each surface mesh, which are in turn used to determine the interpenetrating vertices and contact area. Intersection contours and the contact constraints generated from them are shown in Figure 7.11. `AJL_CONTOUR` supports the `VERTEX_EDGE_PENETRATION` contact method, but usually cannot handle intersections near the edges of an open mesh.

### SIGNED_DISTANCE

Uses a grid-based signed distance field on one mesh to quickly determine the penetrating vertices of the opposite mesh, along with the penetration distance and normals. This is only available for collidable pairs where at least one of the bodies maintains a signed distance grid which can be obtained using `getDistanceGridComp()`. Advantages include speed (sometimes an order of magnitude faster than the other colliders) and the fact that the opposite mesh does not have to be triangular or manifold. However, signed distance fields can (at present) only be computed for fixed meshes, and so at least one colliding body must be rigid. The signed distance field also does not yet yield contact region information, and so the contact method is restricted to `VERTEX_PENETRATION`. Contacts generated from a signed distance field are illustrated in Figure Figure 7.12.

Because the `SIGNED_DISTANCE` collider type currently supports only the `VERTEX_PENETRATION` method, its use between rigid bodies, or low DOF deformable bodies, will generally result in an overconstrained system unless measures are taken as described in Section 7.6.

### 7.4.2.1 Collision meshes and signed distance grids

As described in Section 7.3.4, collidable bodies declare the method `getCollisionMesh()`, which returns a mesh defining the collision surface. This is either used directly, or, for the `SIGNED_DISTANCE` collider, used to compute a signed distance grid which in turn is used to handle collisions.

For `RigidBody` objects, the mesh returned by `getCollisionMesh()` is typically the same as the surface mesh. However, it is possible to change this, by adding additional meshes to the body and modifying the meshes’ `isCollidable` property (see Section 7.3). The collision mesh is then formed as the sum of all polygonal meshes in the body’s `meshes` list whose `isCollidable` property is true.

Collidable bodies also declare the methods `hasDistanceGrid()` and `getDistanceGridComp()`. If the former returns true, then the body has a distance grid and the latter returns a `DistanceGridComp` containing it. A distance grid is a regular 3D grid, with uniformly arranged vertices and cells, that is used to represent a scalar distance field, with distance values stored implicitly at each vertex and interpolated within cells. Distance grids are used by the `SIGNED_DISTANCE`
collider, and by default are generated on-demand, using an automatically chosen resolution and the mesh returned by `getCollisionMesh()` as the surface against which distances are computed.

When used for collision handling, values within a distance grid are interpolated trilinearly within each cell. This means that the effective collision surface is actually the trilinearly interpolated isosurface corresponding to a distance of 0. This surface will differ somewhat from the original surface returned by `getCollisionMesh()`, in a manner that depends on the grid resolution. Consequently, when using the SIGNED_DISTANCE collider, it is important to be able to visualize the trilinear isosurface, and possibly modify it by adjusting the grid resolution. The `DistanceGridComp` returned by `getDistanceGridComp()` exports properties to facilitate both of these requirements, as described in detail in Section 4.5.

### 7.5 Contact rendering

As mentioned in Section 7.2.1, additional collision behavior properties exist to enable and control the rendering of contact artifacts. These include intersection contours, contact points and normals, contact forces and pressures, and penetration depths. The properties that control these are supplemented by generic render properties (Section 4.3) to specify colors, line widths, etc.

By default, contact rendering is disabled. To enable it, one must set the collision manager to be `visible`, which can be done using a code fragment like the following:

```java
RenderProps.setVisible (mechModel.getCollisionManager(), true);
```

`CollisionManager` and `CollisionBehavior` both export contact rendering properties. Setting these in the former controls rendering globally for all collisions, while setting them in the latter controls rendering for the collidable pair associated with the behavior. Some artifacts, like contact normals, forces, and color maps, must be drawn with respect to either the first or second collidable. By default, the first collidable is chosen, but the second collidable can be requested using the `renderingCollidable` property described below. Normals are drawn toward, and forces drawn such that they are acting on, the indicated collidable. For collision behaviors specified using the `setCollisionBehavior()` methods, the first and second collidables correspond to the `collidable0` and `collidable1` arguments. Otherwise, for default collision behaviors, the first collidable will be the one whose collidable bodies have the lowest collidable index (Section 7.3.4).

Properties exported by both `CollisionManager` and `CollisionBehavior` include:

- `renderingCollidable`
Figure 7.12: Contacts generated by a signed distance field. A deformable ellipsoid (red, top) is colliding with a solid prism (cyan, bottom). A signed distance field for the prism (the grid points and normals of which are shown partially by the dark cyan arrows) is used to locate penetrating vertices of the ellipsoid and hence generate contact constraints (yellow lines).

Integer value of either 0 or 1 specifying the collidable for which forces, normals and color maps should be drawn. 0 or 1 selects either the first or second collidable. The default value is 0.

**drawIntersectionContours**

Boolean value requesting drawing of the intersection contours between collidable meshes, using the generic render properties `edgeWidth` and `edgeColor`. The default value is `false`.

**drawIntersectionPoints**

Boolean value requesting drawing of the interpenetrating vertices on each collidable mesh, using the generic render properties `pointStyle`, `pointSize`, `pointRadius`, and `pointColor`. The default value is `false`.

**drawContactNormals**

Boolean value requesting drawing of the normals associated with each contact constraint, using the generic render properties `lineStyle`, `lineRadius`, `lineWidth`, and `lineColor`. The default value is `false`. The length of the normals is controlled by the `contactNormalLen` property of the collision manager, which will be set to an appropriate default value if not set explicitly.

**drawContactForces**

Boolean value requesting drawing of the forces associated with each contact constraint, using the generic render properties `lineStyle`, `lineRadius`, `edgeWidth`, and `edgeColor` (or `lineColor` if `edgeColor` is `null`). The default value is `false`. The forces are drawn as line segments starting at each contact point and running parallel to the contact normal, with a length given by the current contact impulse value multiplied by the `contactForceLenScale` property of the collision manager (which has a default value of 1). The reason for using `edgeWidth` and `edgeColor` instead of `lineWidth` and `lineColor` is to allow the application to set the render properties such that both normals and forces can be visible if both are being rendered at the same time.

**drawFrictionForces**

Boolean value requesting the drawing of the forces associated with each contact's friction force, in the same manner and style as for `drawContactForces`. The default value is `false`. Note that unlike contact forces, friction forces are perpendicular to the contact normal.

**drawColorMap**
An enum of type `CollisionBehavior.ColorMapType` requesting that a color map be drawn over the contact regions showing a scalar value such as penetration depth or contact force pressure. The collidable (0 or 1) onto which the color map is drawn is controlled by the `renderingCollidable` property (described below). The range of values used for generating the map is controlled by the `colorMapRange` property of either the collision manager or the behavior, as described below. The values are mapped onto colors using the `colorMap` property of the collision manager.

Values of `CollisionBehavior.ColorMapType` include:

**NONE**
- The color map is disabled. This is the default value.

**PENETRATION_DEPTH**
- The color map shows penetration depth of one collidable mesh with respect to the other. If one or both collidable meshes are open, then the `colliderType` should be set to `AJL_CONTOUR` (Section 7.2.1).

**CONTACT_PRESSURE**
- The color map shows the contact pressures over the contact region. Contact pressures are determined by examining the forces at the contact constraints and then distributing these over the surrounding faces. This information is most useful and accurate when using vertex penetration contact (Section 7.4.1.2).

The color map itself is drawn as a patch formed from the collidable’s collision mesh, using faces and vertices associated with the collision region. The vertices of the patch are set to colors corresponding to the associated value (e.g., penetration depth or pressure) at that vertex, and the surrounding faces are colored appropriately. The resolution of the color map is thus determined by the resolution of the collision mesh, and so the application must ensure this is high enough to ensure proper results. If the mesh has only a few triangles (e.g., a rectangle with two triangles per face), the color interpolation may be spread over an unreasonably large area. Examples of color map usage are given in Sections 7.5.2 and 7.5.3.

**colorMapRange**
- Composite property of the type `ScalarRange` that controls the range of values used for color map rendering. Subproperties include `interval`, which gives the value range itself, and `updating`, which specifies how this interval should be updated: `FIXED` (no updating), `AUTO_EXPAND` (interval is expanded as values increase or decrease), and `AUTO_FIT` (interval is reset to fit the values at every render step).

When the `colorMapRange` value for a `CollisionBehavior` is set to `null` (which is the default), the `colorMapRange` of the `CollisionManager` is used instead. This has the advantage of ensuring that the same color map range will be used across all collision interactions.

**colorMapInterpolation**
- An enum of type `Renderer.ColorInterpolation` that explicitly specifies how colors in any rendered color map should be interpolated. `RGB` and `HSV` (the default) specify interpolation in RGB and HSV space, respectively. HSV interpolation is the default as it is generally better suited to rendering maps that are purely color-based.

In addition, the following properties are exported by the collision manager:

**contactNormalLen**
- Double value specifying the length with which contact normals should be drawn when `drawContactNormals` is `true`. If unspecified, a default value is calculated by the system.

**contactForceLenScale**
- Double value specifying the scaling factor for contact or friction force vectors when they are drawn in response to `drawContactForces` `drawFrictionForces` being `true`. The default value is 0.

**colorMap**
- Composite property of type `ColorMapBase` specifying the actual color map used for color map rendering. The default value is `HueColorMap`.

Generic render properties within the collision manager can be set in the same way as the visibility, using the `RenderProps` methods presented in Section 4.3.2:
As mentioned above, generic render properties can also be set individually for specific behaviors. This can be done using code fragments like this:

```
CollisionBehavior behav = mechModel.getCollisionBehavior(bodA, bodB);
RenderProps.setLineWidth(behav, 2);
RenderProps.setLineColor(behav, Color.Blue);
```

To access these properties on a read-only basis, one can do

```
RenderProps props = mechModel.getCollisionManager().getRenderProps();
... OR ...
RenderProps props = behav.getRenderProps();
```

Because of the manner in which ArtiSynth handles contacts, rendered contact information may sometimes appear to lag the simulation by one time step. That is because contacts are computed at the end of each time step $i$, and then used to compute the contact forces during the next step $i+1$. The contact information displayed at the end of step $i$ is thus based on contacts detected at the end of step $i-1$, along with contact forces that are computed during step $i$ and used to calculate the updated positions and velocities at the end of that step.

### 7.5.1 Example: rendering normals and contours

A simple model illustrating contact rendering is defined in

```
artisynth.demos.tutorial.BallPlateCollide
```

This shows a ball colliding with a plate, while rendering the resulting contact normals in red and the intersection contours in blue. This demo also allows the user to experiment with compliant contact (Section 7.7.1) by setting compliance and damping properties in a control panel.

![Figure 7.13: BallPlateCollide showing contact normals (red) and collision contour (blue) of the ball colliding with the plate.](image)

The complete source code is shown below:
package artisynth.demos.tutorial;

import java.awt.Color;
import artisynth.core.gui.ControlPanel;
import artisynth.core.mechmodels.CollisionManager;
import artisynth.core.mechmodels.MechModel;
import artisynth.core.mechmodels.RigidBody;
import artisynth.core.workspace.RootModel;
import maspack.matrix.RigidTransform3d;
import maspack.render.RenderProps;
import maspack.render.Renderer;

public class BallPlateCollide extends RootModel {

    public void build (String[] args) {

        // create MechModel and add to RootModel
        MechModel mech = new MechModel("mech");
        addModel (mech);

        // create and add the ball and plate
        RigidBody ball = RigidBody.createIcosahedralSphere("ball", 0.8, 0.1, 1);
        ball.setPose (new RigidTransform3d (0, 0, 2, 0.4, 0.1, 0.1));
        mech.addRigidBody (ball);
        RigidBody plate = RigidBody.createBox("plate", 5, 5, 0.4, 1);
        plate.setDynamic (false);
        mech.addRigidBody (plate);

        // turn on collisions
        mech.setDefaultCollisionBehavior (true, 0.20);

        // make ball transparent so that contacts can be seen more clearly
        RenderProps.setFaceStyle (ball, Renderer.FaceStyle.NONE);
        RenderProps.setShading (ball, Renderer.Shading.NONE);
        RenderProps.setDrawEdges (ball, true);
        RenderProps.setEdgeColor (ball, Color.WHITE);

        // enable rendering of contacts normals and contours
        CollisionManager cm = mech.getCollisionManager();
        RenderProps.setVisible (cm, true);
        RenderProps.setLineWidth (cm, 3);
        RenderProps.setLineColor (cm, Color.RED);
        RenderProps.setEdgeWidth (cm, 3);
        RenderProps.setEdgeColor (cm, Color.BLUE);
        cm.setDrawContactNormals (true);
        cm.setDrawIntersectionContours (true);

        // create a control panel to allow contact regularization to be set
        ControlPanel panel = new ControlPanel();
        panel.addWidget (mech.getCollisionManager (), "compliance");
        panel.addWidget (mech.getCollisionManager (), "damping");
        addControlPanel (panel);
    }
}

The `build()` method starts by creating and adding a `MechModel` in the usual way (lines 19-20). The ball and plate are both created as rigid bodies (lines 22-28), with the ball pose set so that its origin is above the plate at (0, 0, 2) and its orientation is perturbed so that it will not land on the plate symmetrically (line 24). Collisions between the ball and plate are enabled at line 31, with a friction coefficient of 0.2. To allow better visualization of the contacts, the ball is made transparent by disabling the drawing of faces, and instead enabling the drawing of edges in white with no shading (lines 33-37).

Rendering of contacts and normals is established by setting the render properties of the collision manager (lines 39-47).
First, the collision manager is set visible (which it is not by default). Then lines (used to render the contact normals) and edges (used to render to intersection contour) are set to red and blue, each with a pixel width of 3. Drawing of the normals and contour is enabled at lines 46-47.

Lastly, for interactively controlling contact compliance (Section 7.7.1), a control panel is built to allow users to adjust the collision manager’s compliance and damping properties (lines 49-53).

To run this example in ArtiSynth, select All demos > tutorial > BallPlateCollide from the Models menu. When run, the ball will collide with the plate and the contact normals and collision contours will be drawn as shown in Figure 7.13.

To enable compliant contact, set the compliance to a non-zero value. A value of 0.001 (which corresponds to a contact stiffness of 1000) causes the ball to bounce considerably when it lands. To counteract this bouncing, the damping should be set to a non-zero value. Since the ball has a mass of 0.21, formula (7.2) suggests that critical damping (for which \( \zeta = 1 \)) can be achieved with \( D \approx 30 \). This does in fact stop the bouncing. Increasing the compliance to 0.01 results in the ball penetrating the plate by a noticeable amount.

### 7.5.2 Example: rendering a color map

As described above, it is possible to use the `drawColorMap` property of the collision behavior to render a color map over the contact area showing a scalar value such as penetration depth or contact pressure. A simple example of this is defined in

```
    artisynth.demos.tutorial.PenetrationRender
```

which sets `drawColorMap` to `PENETRATION_DEPTH` in order to display the penetration depth of one hemispherical mesh with respect to another.

As mentioned above, proper results require that the collision mesh for the collidable on which the map is being drawn has a sufficiently high resolution.

![PenetrationRender](image)

Figure 7.14: PenetrationRender showing the penetration depth of the bottom mesh with respect to the top, with red indicating greater depth. A translation dragger fixture at the top is being used to move the top mesh around, while the penetration range and associated colors are displayed on the color bar at the right.

The complete source code is shown below:

```java
1 package artisynth.demos.tutorial;
2
3 import java.awt.Color;
4
```
import maspack.geometry.PolygonalMesh;
import maspack.geometry.MeshFactory;
import maspack.matrix.RigidTransform3d;
import maspack.render.*;
import maspack.render.Renderer.FaceStyle;
import maspack.render.Renderer.Shading;
import artisynth.core.mechmodels.*;
import artisynth.core.mechmodels.CollisionManager.ColliderType;
import artisynth.core.mechmodels.CollisionBehavior.ColorMapType;
import artisynth.core.util.ScalarRange;
import artisynth.core.workspace.RootModel;
import artisynth.core.renderables.ColorBar;
import maspack.render.color.JetColorMap;

public class PenetrationRender extends RootModel {

    // Convenience method for creating colors from [0-255] RGB values
    private static Color createColor(int r, int g, int b) {
        return new Color(r/255.0f, g/255.0f, b/255.0f);
    }

    private static Color CREAM = createColor(255, 255, 200);
    private static Color GOLD = createColor(255, 150, 0);

    // Creates and returns a rigid body built from a hemispherical mesh. The
    // body is centered at the origin, has a radius of `rad`, and the z axis is
    // scaled by `zscale`.
    RigidBody createHemiBody(MechModel mech, String name, double rad, double zscale, boolean flipMesh) {
        PolygonalMesh mesh = MeshFactory.createHemisphere(
            rad, /* slices=*/20, /* levels=*/10);
        mesh.scale(1, 1, zscale); // scale mesh in the z direction
        if (flipMesh) {
            // flip upside down is requested
            mesh.transform(new RigidTransform3d(0, 0, 0, 0, 0, Math.PI));
        }
        RigidBody body = RigidBody.createFromMesh {
            name, mesh, /* density=*/'1000, /* scale=*/1.0);
        mech.addRigidBody (body);
        body.setDynamic (false); // body is only parametrically controlled
        return body;
    }

    // Creates and returns a ColorBar renderable object
    public ColorBar createColorBar() {
        ColorBar cbar = new ColorBar();
        cbar.setName("colorBar");
        cbar.setNumberFormat("%.2f"); // 2 decimal places
        cbar populateLabels(0.0, 1.0, 10); // Start with range [0,1], 10 ticks
        cbar setLocation(-100, 0.1, 20, 0.8);
        cbar setTextColor (Color.WHITE);
        addRenderable(cbar); // add to root model’s renderables
        return cbar;
    }

    public void build(String[] args) {
        MechModel mech = new MechModel("mech");
        addModel (mech);
        // create first body and set its rendering properties
        RigidBody body0 = createHemiBody (mech, "body0", 2, -0.5, false);
        RenderProps.setFaceStyle (body0, FaceStyle.FRONT_AND_BACK);
        RenderProps.setFaceColor (body0, CREAM);
    }
}
// create second body and set its pose and rendering properties
RigidBody body1 = createHemiBody(mech, "body1", 1, 2.0, true);
body1.setPose(new RigidTransform3d(0, 0, 0.75));
RenderProps.setFaceStyle(body1, FaceStyle.NONE); // set up
RenderProps.setShading(body1, Shading.NONE); // wireframe
RenderProps.setDrawEdges(body1, true); // rendering
RenderProps.setEdgeColor(body1, GOLD);

// create and set a collision behavior between body0 and body1, and make
// collisions INACTIVE since we only care about graphical display
CollisionBehavior behav = new CollisionBehavior(true, 0);
behav.setMethod(CollisionBehavior.Method.INACTIVE);
behav.setDrawColorMap(ColorMapType.PENETRATION_DEPTH);
behav.setRenderingCollidable(1); // show penetration of mesh 0
mech.setCollisionBehavior(body0, body1, behav);

CollisionManager cm = mech.getCollisionManager();
// works better with open meshes if AJL_CONTOUR is selected
cm.setColliderType(ColliderType.AJL_CONTOUR);
// set other rendering properties in the collision manager:
RenderProps.setVisible(cm, true); // enable collision rendering
RenderProps.setDrawIntersectionContours(true); // draw contours ...
RenderProps.setLineWidth(cm, 3); // with a line width of 3
RenderProps.setEdgeColor(cm, Color.BLUE); // and a blue color
// create a custom color map for rendering the penetration depth
JetColorMap map = new JetColorMap();
map.setColorArray(new Color[]{
    CREAM, // no penetration
    createColor(255, 204, 153),
    createColor(255, 153, 102),
    createColor(255, 102, 51),
    createColor(240, 0, 0), // most penetration
});
cm.setColorMap(map);
// set color map range to "auto fit".
ScalarRange range = cm.getColorMapRange();
// create a separate color bar to show depth values associated with the
// color map
ColorBar cbar = createColorBar();
cbar.setColorMap(map);

public void prerender(RenderList list) {
    super.prerender(list); // call the regular prerender method first
    // Update the color bar labels based on the collision manager’s
    // color map range that was updated in super.prerender().
    // Object references are obtained by name using ‘findComponent’. This is
    // more robust than using class member variables, since the latter will
    // be lost if we save and restore this model from a file.
    ColorBar cbar = (ColorBar) renderables().get("colorBar");
    MechModel mech = (MechModel) findComponent("models/mech");
    RigidBody body0 = (RigidBody) mech.findComponent("rigidBodies/body0");
    RigidBody body1 = (RigidBody) mech.findComponent("rigidBodies/body1");
    CollisionManager cm = mech.getCollisionManager();
    ScalarRange range = cm.getColorMapRange();
    cbar.updateLabels(0, 1000*range.getUpperBound());
}
To improve visibility, the example uses two rigid bodies, each created from an open hemispherical mesh using the method `createHemiBody` (lines 32-47). Because this example is strictly graphical, the bodies are set to be non-dynamic so that they can be moved around using the viewer’s graphical dragger fixtures (see the section “Transformer Tools” in the ArtiSynth User Interface Guide). Rendering properties for each body are set at lines 67-68 and 73-76, with the top body being rendered as a wireframe to improve visibility.

Lines 80-84 create and set a collision behavior between the two bodies with the `drawColorMap` property set to `PENETRATION_DEPTH`. Because for this example we want to show only the penetration and don’t want a collision response, we set the contact method to be `Method.INACTIVE`. At line 88, the collider type used by the collision manager is set to `ColliderType.AJL_CONTOUR`, since this provides more reliable penetration calculations for open meshes. Other rendering properties are set for the collision manager at lines 90-107, including a custom color map that varies between `CREAM` (the color of the mesh) for no penetration and dark red for maximum penetration. The updating of the color map range in the collision manager is set to `AUTO_FIT` so that it will be recomputed at every time step. (Since the collision manager’s color map range is set to “auto fit” by default, this is shown for illustrative purposes only. It is also possible to override the collision manager’s color map range by setting the `colorMapRange` property in specific collision behaviors.)

At line 111, a color bar is created and added to the scene, using the method `createColorBar` (lines 50-59), to explicitly show the depth that corresponds to the different colors. The color bar is given the same color map that is used to render the depth. Since the depth range is updated every time step, it is also necessary to update the corresponding labels in the color bar. This is done by overriding the root model’s `prerender` method (lines 115-132), where we obtain the color map range for the collision manager and use it to update the color bar labels. (Note that `super.prerender(list)` is called first, since the color map ranges are updated there.) References to the color bar, `MechModel`, and bodies are obtained using the `CompositeComponent` methods `get()` and `findComponent()`. This is more robust that storing these references in member variables, since the latter would be lost if the model is saved to and reloaded from a file.

To run this example in ArtiSynth, select `All demos > tutorial > PenetrationRender` from the Models menu. When run, the meshes will collide and render the penetration depth of the bottom mesh, as shown in Figure 7.14.

When defining the color map for rendering (lines 99-108 in the example), it is recommended that the color corresponding to zero be set to the face color of the collidable mesh. This will allow the color map to blend properly to the regular mesh color.

7.5.3 Example: rendering contact pressures

Color map rendering can also be used to render contact pressures, which can be particularly useful for FEM models. A simple example is defined in

`artisynth.demos.tutorial.ContactPressureRender`

which sets the `drawColorMap` property of the collision behavior to `CONTACT_PRESSURE` in order to display a color map of the contact pressure. The example is similar to that of Section 7.5.2, which shows how to render penetration depth.

The caveats about color map rendering described in Section 7.5 apply. Pressure rendering is most useful and accurate when using vertex penetration contact. The resolution of the color map is limited by the resolution of the collision mesh for the collidable on which the map is drawn, and so the application should ensure that this is sufficiently fine. Also, to allow the map to blend properly with the rest of the collidable, the color corresponding to zero pressure should match the default face color for the collidable.

The complete source code is shown below:

```java
package artisynth.demos.tutorial;

import java.awt.Color;
import artisynth.core.femmodels.FemFactory;
import artisynth.core.femmodels.FemModel.SurfaceRender;
import artisynth.core.femmodels.FemModel3d;
import artisynth.core.femmodels.FemNode3d;
```
Figure 7.15: ContactPressureRender showing the contact pressure as a spherical FEM model falls onto an FEM sheet. The color map is drawn on the sheet model, with redder values indicating greater pressure.

```java
import artisynth.core.materials.LinearMaterial;
import artisynth.core.mechmodels.CollisionBehavior;
import artisynth.core.mechmodels.CollisionManager;
import artisynth.core.mechmodels.ColorMapType;
import artisynth.core.mechmodels.CollisionManager;
import artisynth.core.renderables.ColorBar;
import artisynth.core.util.ScalarRange;
import artisynth.core.workspace.RootModel;
import maspack.matrix.RigidTransform3d;
import maspack.render.RenderList;
import maspack.render.RenderProps;
import maspack.render.color.JetColorMap;

public class ContactPressureRender extends RootModel {
    double density = 1000;
    double EPS = 1e-10;

    // Convenience method for creating colors from [0-255] RGB values
    private static Color createColor(int r, int g, int b) {
        return new Color(r/255.0f, g/255.0f, b/255.0f);
    }

    private static Color CREAM = createColor(255, 255, 200);
    private static Color BLUE_GRAY = createColor(153, 153, 255);

    // Creates and returns a ColorBar renderable object
    public ColorBar createColorBar() {
        ColorBar cbar = new ColorBar();
        cbar.setName("colorBar");
        cbar.setNumberFormat("%.2f"); // 2 decimal places
        cbar.populateLabels(0.0, 1.0, 10); // Start with range [0,1], 10 ticks
        cbar.setLocation(-100, 0.1, 20, 0.8);
        cbar.setTextColor(Color.WHITE);
        addRenderable(cbar); // add to root model’s renderables
        return cbar;
    }

    public void build(String[] args) {
        MechModel mech = new MechModel("mech");
    }
```
To begin, the demo creates two FEM models: a spherical ball (lines 52-56) and a rectangular sheet (lines 59-66), and then fixes the end nodes of the sheet (lines 69-74). Surface rendering is enabled for the sheet (line 65), but not for the ball, in order to improve the visibility of the color map.
Lines 77-81 create and set a collision behavior between the two models, with the `drawColorMap` property set to `CONTACT_PRESSURE`. Because for this example we want the color map to be drawn on the second collidable (the sheet), we set the `setColorMapCollidable` property to 1 (line 79); otherwise, the default value of 0 would cause the color map to be drawn on the first collidable (the ball). (Alternatively, we could have simply defined the collision behavior as being between the surface and the ball instead of the ball and the sheet.) The color map range is explicitly set to lie between \([0, 15000]\) (line 80); this is in contrast to the example in Section 7.5.2, where the range is auto-updated on each step. The color range is also set explicitly in the behavior, but if multiple objects were colliding it would likely be preferable to set it in the collision manager (with the behavior’s value left as null) to ensure a uniform render range across all collisions. Other rendering properties are set for the collision manager at lines 83-97, including a custom color map that varies between CREAM (the color of the mesh) for no pressure and dark red for maximum pressure.

At line 100, a color bar is created and added to the scene, using the method `createColorBar()` (lines 36-45), to explicitly show the pressure that corresponds to the different colors. The color bar is given the same color map and value range used to render the pressure. Finally, default face and line colors for all components in the model are set at lines 105-106.

To run this example in ArtiSynth, select `All demos > tutorial > ContactPressureRender` from the Models menu. When run, the FEM models will collide and render the contact pressure on the sheet, as shown in Figure 7.15.

### 7.6 Overconstrained contact

When contact occurs between collidable bodies, the system creates a set of contact constraints to handle the collision and passes these to the physics solver (Section 1.2). When one or both of the contacting collidables are FEM models, the number of constraints is usually less than the DOF count of the collidables. However, if the both collidables are rigid bodies, then the number of constraints often exceeds the DOF count, a situation which is referred to as redundant contact. Redundant contact can also occur in other collision situations, such those involving embedded meshes (Section 6.3.2) when the mesh has a finer resolution than the embedding FEM, or skinned meshes (Chapter 9) when the number of contacts exceeds the DOF count of the underlying master bodies.

Redundant contact is not usually a problem in collisions between rigid bodies, because by default ArtiSynth handles these using `CONTOUR_REGION` contact (Section 7.4.1), for which contacts are supplied to the physics solver as unilateral constraints which are solved using complementarity techniques that can handle redundant constraints. However, as mentioned in Section 7.4.1.2, the default implementation for vertex penetration contact is to present the contacts to the physics solver as bilateral constraints (Section 1.2), which are removed between simulation steps. This results in much faster simulation times, and usually works well in the case of FEM models, where the DOF count almost always exceeds the number of constraints. However, when vertex penetration contact is employed between rigid bodies, and sometimes in other situations as described above, redundant contact can arise, and then the use of bilateral constraints results in overconstrained contact for which the solver has difficulty finding a proper solution.

A common indicator of overconstrained contact is for an error message to appear in ArtiSynth’s console output, which typically looks like this:

```
Pardiso: num perturbed pivots=12
```

The simulation may also go unstable.

At present there are three general ways to manage overconstrained contact:

1. Requiring the system to use unilateral constraints for vertex penetration contact. This can be done by setting the property `bilateralVertexContact` (Section 7.2.1) to `false` in either the collision manager or behavior, but the resulting hit to computational performance may be large.

2. Constraint reduction, described in Section 7.6.1.

3. Regularizing the contact, using one of the methods of Section 7.7.

When using the new implicit friction integration feature (Section 7.9.4), if redundant contacts arise, it is necessary to regularize both the contact constraints, as per Section 7.7, as well as the friction constraints, by setting the `stictionCreep` property of either the collision manager or behavior to a non-zero value, as described in Section 7.2.1.
7.6.1 Constraint reduction

Constraint reduction involves having the collision manager explicitly try to reduce the number of collision constraints to match the available DOFs, and is enabled by setting the `reduceConstraints` property for either the collision manager or the `CollisionBehavior` for a particular collision pair to `true`. The default value of `reduceConstraints` is `false`. As with all properties, it can be set interactively in the GUI, or in code using the property’s accessor methods,

```java
boolean getReduceContraints()
void setReduceContraints (boolean enable)
```

as illustrated by the following code fragments:

```java
MechModel mech;
...

// enable constraint reduction for all collisions:
mech.getCollisionManager().setReduceContraints (true);

// enable constraint reduction for collisions between bodyA and bodyB:
CollisionBehavior behav = 
  mech.setCollisionBehavior (bodyA, bodyB, true);
behav.setReduceContraints (true);
```

7.7 Contact regularization

Contact regularization is a technique in which contact constraints are made “soft” by adding compliance and damping. This means that they no longer remove DOFs from the system, and so overconstraining cannot occur, but contact penetration is no longer strictly enforced and there may be an increase in the computation time required for the simulation.

7.7.1 Compliant contact

Compliant contact is a simple form of regularization that is analogous to that used for joints and connectors, discussed in Section 3.3.8. Compliance and damping parameters $C$ and $D$ can be specified between any two collidables, with $C$ being the inverse of a corresponding contact stiffness $K = 1/C$, such the contact forces $f_c$ acting along each contact normal are given by

$$f_c = -Kd - D\dot{d}, \quad (7.1)$$

where $d$ is the contact penetration distance and is negative when penetration occurs. Contact forces act to push the contacting bodies apart, so if body A is penetrating body B and the contact normal $n$ is directed toward A, the forces acting on A and B at the contact point will be $f_c n$ and $-f_c n$, respectively. Since $C$ is the inverse of the contact stiffness $K$, a value of $C = 0$ (the default) implies “infinitely” stiff contact constraints. Compliance is enabled whenever the compliance is set to a value greater than 0, with stiffness decreasing as compliance increases.

Compliance can be enabled by setting the compliance and damping properties of either the collision manager or the `CollisionBehavior` for a specific collision pair; these properties correspond to $C$ and $D$ in (7.1). While it is not required to set the damping property, it is usually desirable to set it to a value that approximates critical damping in order to prevent “bouncing” contact behavior. Compliance and damping can be set in the GUI by editing the properties of either the collision manager or a specific collision behavior, or set in code using the accessor methods

```java
double getCompliance()
double setCompliance (double c)

double getDamping()
double setDamping (double d)
```

as is illustrated by the following code fragments:

```java
MechModel mech;
double compliance;
double damping;
```
... determine compliance and damping values ...

```java
// set damping and compliance for all collisions:
mech.getCollisionManager().setCompliance (compliance);
mech.getCollisionManager().setDamping (damping);

// enable compliance and damping between bodyA and bodyB:
CollisionBehavior behav = mech.setCollisionBehavior (bodyA, bodyB, true);
behav.setCompliance (compliance);
behav.setDamping (damping);
```

An important question is what values to choose for compliance and damping. At present, there is no automatic way to determine these, and so some experimental parameter tuning is often necessary. With regard to the contact stiffness $K$, appropriate values will depend on the desired maximum penetration depth and other loadings acting on the body. The mass of the collidable bodies may also be relevant if one wants to control how fast the contact stabilizes. (The mass of every CollidableBody can be determined by its getMass() method.) Since $K$ acts on a per-contact basis, the resulting total force will increase with the number of contacts. Once $K$ is determined, the compliance value $C$ is simply the inverse: $C = 1/K$.

Given $K$, the damping $D$ can then be estimated based on the desired damping ratio $\zeta$, using the formula

$$D = 2\zeta \sqrt{KM}$$

(7.2)

where $M$ is the combined mass of the collidable bodies (or the mass of one body if the other is non-dynamic). Typically, the desired damping will be close to critical damping, for which $\zeta = 1$.

An example that allows a user to play with contact regularization is given in Section 7.5.1.

### 7.7.2 Contact force behaviors

When regularizing contact through compliance, as described in Section 7.6, the forces $f_c$ at each contact are explicitly determined as per (7.1). As a broader alternative to this, ArtiSynth allows applications to specify a contact force behavior, which allows $f_c$ to be computed as a more generalized function of $d$:

$$f_c = -F(d) - D(d) \dot{d},$$

(7.3)

where $F(d)$ and $D(d)$ are functions returning the elastic force and the damping coefficient. The corresponding compliance is then a local function of $d$ given by $C(d) = 1/F'(d)$ (which is the inverse of the local stiffness).

Because $d$ is negative during contact, $F(d)$ should also be negative. $D(d)$, on the other hand, should be positive, so that the damping acts to reduce $d$.

Contact force behaviors are implemented as subclasses of the abstract class ContactForceBehavior. To enable them, the application sets the forceBehavior property of the CollisionBehavior controlling the contact interaction, using the method

```java
setForceBehavior (ContactForceBehavior behav)
```

A ContactForceBehavior must implement the method `computeResponse()` to compute the contact force, compliance and damping at each contact:

```java
void computeResponse (double[] fres, double dist, ContactPoint cpnt0, ContactPoint cpnt1,
                      Vector3d nrml, double contactArea, int flags);
```

The arguments to this method supply specific information about the contact (see Figure 7.16):

- `fres`:
  An array of length three that returns the contact force, compliance and damping in its first, second and third entries. The contact force is the $F(d)$ shown in (7.3), the compliance is $1/F'(d)$, and the damping is $D(d)$. 

Figure 7.16: Highly magnified schematic of a single vertex penetration contact between body A (green) and body B (blue). cpnt0 is the point on A penetrating B, while cpnt1 is the corresponding nearest point on the surface of B. The contact normal \( n \) faces outward from the surface of B towards A, and the penetration distance \( d \) is the (negative valued) distance along \( n \) from cpnt1 to cpnt0.

**dist**

The penetration distance \( d \) (the value of which is negative).

**cpnt0, cpnt1**

ContactPoint objects containing information about the two points associated with the contact (Figure 7.16), including each point’s position (returned by `getPoint()`) and associated vertices and weights (returned by `numVertices()`, `getVertices()`, and `getWeights()`).

**nrml**

The contact normal \( n \).

**contactArea**

Average area per contact, computed by dividing the total area of the contact region by the number of penetrating vertices. This information is only available if the `colliderType` property of the collision behavior is set to `AJL_CONTOUR` (Section 7.2.1); otherwise, `contactArea` will be set to -1.

A collision force behavior may use its `computeResponse()` method to calculate the force, compliance and damping in any desired way. As per equation (7.3), these quantities are functions of the penetration distance \( d \), supplied by the argument `dist`.

A very simple instance of `ContactForceBehavior` that just recreates the contact compliance of (7.1) is shown below:

```java
class SimpleCompliantContact extends ContactForceBehavior {
    double myStiffness = 10000.0; // contact stiffness
    double myDamping = 10.0;

    public void computeResponse (double[] fres, double dist, ContactPoint cpnt0, ContactPoint cpnt1,
                                  Vector3d nrml, double regionArea, int flags) {
        fres[0] = dist*myStiffness; // contact force
        fres[1] = 1/myStiffness;     // compliance is inverse of stiffness
        fres[2] = myDamping;        // damping
    }
}
```

This example uses only the penetration distance information supplied by `dist`, dividing it by the compliance to determine the contact force. Since `dist` is negative when the collidables are interpenetrating, the returned contact force will also be negative.
7.7.2.1 Computing forces based on pressure

Often, such as when implementing elastic foundation contact (Section 7.7.3), the contact force behavior is expressed in terms of pressure, given as a function of $d$:

$$p = G(d).$$

(7.4)

In that case, the contact force $F(d)$ is determined by multiplying $G(d)$ by the local area $A$ associated with the contact:

$$F(d) = AG(d).$$

(7.5)

Since $d$ and $F(d)$ are both assumed to be negative during contact, we will assume here that $G(d)$ is negative as well.

To find the area $A$ within the `computeResponse()` method, one has two options:

A) Compute an estimate of the local contact area surrounding the contact, as discussed below. This may be the more appropriate option if the colliding mesh vertices are not uniformly spaced, or if the `contactArea` argument is undefined because the `AJL_CONTOUR` collider (Section 7.4.2) is not being used.

B) Use the `contactArea` argument if it is is defined. This gives an accurate estimate of the per-contact area on the assumption that the colliding mesh vertices are uniformly distributed. `contactArea` will be defined when using the `AJL_CONTOUR` collider (Section 7.4.2); otherwise, it will be set to -1.

For option (A), `ContactForceBehavior` provides the convenience method `computeContactArea(cpnt0,cpnt1,nrml)`, which estimates the local contact area given the contact points and normal. If the above example of `computeResponse()` is modified so that the stiffness parameter controls pressure instead of force, then we obtain the following:

```java
public void computeResponse {
    double[] fres, double dist, ContactPoint cpnt0, ContactPoint cpnt1,
    Vector3d nrml, double regionArea, int flags) {

    // assume stiffness determines pressure instead of force,
    // so it needs to be scaled by the local contact area:
    double K = myStiffness * computeContactArea (cpnt0, cpnt1, nrml);

    fres[0] = dist*K;     // contact force
    fres[1] = 1/K;        // compliance
    fres[2] = myDamping;  // damping
}
```

`computeContactArea()` only works for vertex penetration contact (Section 7.4.1.2), and will return -1 otherwise. That’s because it needs the contact points’ vertex information to compute the area. In particular, for vertex penetration contact, it associates the vertex with 1/3 of the area of each of its surrounding faces.

When computing contact forces, care must also be taken to consider whether the vertex penetrations are being computed for both colliding surfaces (two-way contact, Section 7.4.1.2). This means that forces are essentially being computed twice - once for each side of the penetrating surface. For forces based on pressure, the resulting contact forces should then be halved. To determine when two-way contact is in effect, the `flags` argument can be examined for the flag `TWO_WAY_CONTACT`:

```java
public void computeResponse {
    double[] fres, double dist, ContactPoint cpnt0, ContactPoint cpnt1,
    Vector3d nrml, double regionArea, int flags) {

    double K = myStiffness * computeContactArea (cpnt0, cpnt1, nrml);
    if (((flags & TWO_WAY_CONTACT) != 0) {
        K *= 0.5;     // divide force response by 2
    }
    fres[0] = dist*K;     // contact force
    fres[1] = 1/K;        // compliance
    fres[2] = myDamping;  // damping
}
```
7.7.3 Elastic foundation contact

ArtiSynth supplies a built-in contact force behavior, `LinearElasticContact`, that implements elastic foundation contact (EFC) based on a linear material law as described in [2]. By default, the contact pressures are computed from

\[
G(d) = -\frac{(1 - \nu)E}{(1 + \nu)(1 - 2\nu)} \ln \left(1 + \frac{d}{h}\right), \tag{7.6}
\]

where \( E \) is Young’s modulus, \( \nu \) is Poisson’s ratio, \( d \) is the contact penetration distance, and \( h \) is the foundation layer thickness. (Note that both \( G(d) \) and \( d \) are negated relative to the formulation in [2] because we assume \( d < 0 \) during contact.) The use of the \( \ln() \) function helps ensure that contact does not penetrate the foundation layer. However, to ensure robustness in case contact does penetrate the layer (or comes close to doing so), the pressure relationship is linearized (with a steep slope) whenever \( d < h(r - 1) \), where \( r \) is the minimum thickness ratio controlled by the `LinearElasticContact` property `minThicknessRatio`.

Alternatively, if a strictly linear pressure relationship is desired, this can be achieved by setting the property `useLogDistance` to `false`. The pressure relationship then becomes

\[
G(d) = \frac{(1 - \nu)E}{(1 + \nu)(1 - 2\nu)} \frac{d}{h}, \tag{7.7}
\]

where again \( G(d) \) will be negative when \( d < 0 \).

Damping forces \( f_d = -D(d)\dot{d} \) can be computed in one of two ways: either with \( D(d) \) set to a constant \( C \) such that

\[
f_d = -C\dot{d}, \tag{7.8}
\]

or with \( D(d) \) set to a multiple of \( C \) and the magnitude of the elastic contact force \( |F(d)| \), such that

\[
f_d = -C|F(d)|\dot{d}. \tag{7.9}
\]

The latter is compatible with the EFC damping used by OpenSim (equation 24 in [18]).

Elastic foundation contact assumes the use of vertex penetration contact (Section 7.4.1) to achieve correct results; otherwise, a runtime error may result.

`LinearElasticContact` exports several properties to control the force response described above:

- **younsgsModulus**
  
  \( E \) in equations (7.7) and (7.6).

- **poissonsRatio**
  
  \( \nu \) in equations (7.7) and (7.6).

- **thickness**
  
  \( h \) in equations (7.7) and (7.6).

- **minThicknessRatio**
  
  \( r \) value such that (7.6) is linearized if \( d < h(r - 1) \). The default value is 0.01.

- **dampingFactor**
  
  \( C \) in equations (7.8) and (7.9).

- **dampingMethod**
  
  An enum of type `ElasticContactBase.DampingMethod`, for which `DIRECT` and `FORCE` cause damping to be implemented using (7.8) and (7.9), respectively. The default value is `DIRECT`.

- **useLogDistance**
  
  A boolean, which if `true` causes (7.6) to be used instead of (7.7). The default value is `true`.

- **useLocalContactArea**
  
  A boolean, which if `true` causes contact areas to be estimated from the vertex of the penetrating contact point (option A in Section 7.7.2.1). The default value is `true`. 

7.7.4 Example: elastic foundation contact of a ball in a bowl

A simple example of elastic foundation contact is defined by the model

artisynth.demos.tutorial.ElasticFoundationContact

This implements EFC between a spherical ball and a bowl into which it is dropped. Pressure rendering is enabled (Section 7.5.3), and the bowl is made transparent, allowing the contact pressure to be viewed as the ball moves about.

![ElasticFoundationContact showing contact pressure between the ball and bowl during contact.](image)

The source code for the build method is shown below:

```java
public void build (String[] args) throws IOException {
    MechModel mech = new MechModel("mech");
    mech.setGravity (0, 0, -9.8);
    addModel (mech);

    // read in the mesh for the bowl
    PolygonalMesh mesh = new PolygonalMesh {
        PathFinder.getSourceRelativePath {
            ElasticFoundationContact.class, "data/bowl.obj"};

    // create the bowl from the mesh and make it non-dynamic
    RigidBody bowl = 
        RigidBody.createFromMesh {
            "bowl", mesh, /*density=*/1000.0, /*scale=*/1.0};
    mech.addRigidBody (bowl);
    bowl.setDynamic (false);

    // create another spherical mesh to define the ball
    mesh = MeshFactory.createICosahedralSphere (0.7, 3);

    // create the ball from the mesh
    RigidBody ball = 
        RigidBody.createFromMesh {
            "ball", mesh, /*density=*/1000.0, /*scale=*/1.0};

    // move the ball into an appropriate "drop" position
    ball.setPose (new RigidTransform3d (0.1, 0, 0));
    mech.addRigidBody (ball);

    // Create a collision behavior that uses EFC. Set friction
    // to 0.1 so that the ball will actually roll.
    CollisionBehavior behav = new CollisionBehavior (true, 0.1);
    behav.setMethod (Method.VERTEX_PENETRATION); // needed for EFC
```
The demo first creates the ball and the bowl as rigid bodies (lines 6-26), with the bowl defined by a mesh read from the file data/bowl.obj relative to the demo source directory, and the ball defined as an icosahedral sphere. The bowl is fixed in place (bowl.setDynamic(false)), and the ball is positioned at a suitable location for dropping into the bowl when simulation begins.

Next, a collision behavior is created, with its collision method property set to VERTEX_PENETRATION (as required for EFC), and a friction coefficient of 0.1 (to ensure that the ball will actually roll) (lines 28-31). A LinearElasticContact is then constructed, with \( E = 10^5 \), \( \nu = 0.4 \), damping factor 0.1, and thickness 0.1, and added to the collision behavior using setForceBehavior() (lines 32-26). The behavior is then used to enable contact between the ball and bowl (line 39).

Lines 41-49 set up rendering properties: the collision manager is made visible, with color map rendering set to CONTACT_PRESSURE, and the ball and bowl are set to be rendered in pale blue and green, with the bowl rendered using only mesh edges so as to make it transparent.

Finally, a control panel is created allowing the user to adjust properties of the contact force behavior and the pressure rendering range (lines 51-57).

To run this example in ArtiSynth, select All demos > tutorial > ElasticFoundationContact from the Models menu. Running the model will cause the ball to drop into the bowl, with the resulting contact pressures rendered as in Figure 7.17.

### 7.7.5 Example: binding EFC properties to fields

It is possible to bind certain EFC material properties to a field, for situations where these properties vary over the surface of the contacting meshes. Properties of LinearElasticContact that can be bound to (scalar) fields include YoungsModulus and thickness, using the methods

```java
setYoungsModulusField {ScalarFieldComponent fcomp}
setThicknessField {ScalarFieldComponent fcomp}
```
Most typically, the properties will be bound to a mesh field (Section 6.10.3) defined with respect to one of the colliding meshes.

When determining the value of either *youngsModulus* or *thickness* within its *computeResponse()* method, *LinearElasticContact* checks to see if the property is bound to a field. If it is, the method first checks if the field is a mesh field associated with the meshes of either *cpnt0* or *cpnt1*, and if so, extracts the value using the vertex information of the associated contact point. Otherwise, the value is extracted from the field using the position of *cpnt0*.

A simple example of binding the *thickness* property to a field is given by

```
artisynth.demos.tutorial.VariableElasticContact
```

This implements EFC between a beveled cylinder and a plate on which it is resting. The plate is made transparent, allowing the contact pressure to be viewed from below.

![VariableElasticContact showing the contact pressure decreasing radially from the cylinder center, as the EFC thickness property increases due to its field binding (left). Without the field binding, the thickness and the pressure are constant across the cylinder bottom (right).](image)

Figure 7.18: VariableElasticContact showing the contact pressure decreasing radially from the cylinder center, as the EFC thickness property increases due to its field binding (left). Without the field binding, the thickness and the pressure are constant across the cylinder bottom (right).

The model’s code is similar to that for *ElasticFoundationContact* (Section 7.7.4), except that the collidables are different and the EFC thickness property is bound to a field. The source code for the first part of the build method is shown below:

```java
public void build (String[] args) throws IOException {
   MechModel mech = new MechModel ("mech");
   mech.setGravity (0, 0, -9.8);
   addModel (mech);

   // read in the mesh for the top cylinder
   PolygonalMesh mesh = new PolygonalMesh (
      PathFinder.getSourceRelativePath (new VariableElasticContact.class, "data/beveledCylinder.obj"));

   // create a cylinder from the mesh
   RigidBody cylinder = RigidBody.createFromMesh (new "cylinder", mesh, /* density=*/1000.0, /* scale=*/1.0);
   mech.addRigidBody (cylinder);

   // create a plate from a box mesh and make it non-dynamic
   RigidBody plate =
      RigidBody.createBox (new "plate", 1.5, 1.5, 0.5, 10, 10, 4, /* density=*/1000.0, false);
   plate.setPose (new RigidTransform3d (0, 0, -0.75));
```

First, the cylinder and the plate are created as rigid bodies (lines 6-23), with the cylinder defined by a mesh read from the file data/beveledCylinder.obj relative to the demo source directory. The plate is fixed in place and positioned so that it is directly under the cylinder.

Next, the contact method is set to VERTEX_PENETRATION (as required for EFC), this time by setting it for all collisions in the collision manager, and a LinearElasticContact is constructed, with \( E = 10^5 \), \( \nu = 0.4 \), damping factor 0.1, and thickness 0.1 (lines 25-32). Then in lines 34-43, a ScalarMeshField is created for the cylinder surface mesh, with vertex values set so that the field defines a thickness value \( h \) that increases with the radial distance \( r \) from the cylinder axis according to:

\[
h = \text{thickness} \times (1 + r)
\]

This field is then added to the fields list of the MechModel and the EFC thickness property is bound to it (lines 44-46).

Lastly, a collision behavior is created, with the EFC added to it as a force behavior, and used to enable cylinder/plate collisions (lines 48-53).

> It is important that EFC properties are bound to fields before the behavior method setForceBehavior() is called, because that method copies the force behavior and so subsequent field bindings would not be noticed. If for some reason the bindings must be made after the call, they should be applied to the copied force behavior, which can be obtained using getForceBehavior().

To run this example in ArtiSynth, select All demos > tutorial > VariableElasticContact from the Models menu. Running the model will result in contact pressures seen in the left of Figure 7.18, with the pressures decreasing radially in response to the increasing thickness. With no field binding, the pressures are constant across the cylinder bottom (Figure 7.18, right).
7.8 Monitoring collisions

Sometimes, an application may wish to know details about the collision interactions between a specific collidable and one or more other collidables. These details may include items such as contact forces or the penetration regions between the opposing meshes. This section describes ways in which these details can be observed through the use of collision responses.

7.8.1 Collision responses

Applications can track collision information by creating `CollisionResponse` components for the collidables in question and adding them to the `MechModel` using `setCollisionResponse()`:

```java
CollisionResponse resp = new CollisionResponse();
mech.setCollisionResponse (collidable0, collidable1, resp);
```

An alternate version of `setCollisionResponse()` creates the response component internally and returns it:

```java
CollisionResponse resp = mech.setCollisionResponse (collidable0, collidable1);
```

During every simulation step, the `MechModel` will update its response components to reflect the current collision conditions between the associated collidables.

The first collidable specified for a collision response must be a specific collidable component, while the second may be either another collidable or a group of collidables represented by a `Collidable.Group`:

```java
CollisionResponse r0, r1, r2;
RigidBody bodA;
FemModel3d femB;

// collision information between bodA and femB only:
r0 = setCollisionResponse (bodA, femB);
// collision information between femB and all rigid bodies:
r1 = setCollisionResponse (femB, Collidable.Rigid);
// collision information between femB and all bodies and self-collisions:
r2 = setCollisionResponse (femB, Collidable.All);
```

When a compound collidable is specified, the response component will collect collision information for all its subcollidables.

As as with collision behaviors, the same response cannot be added to an application twice:

```java
CollisionResponse resp = new CollisionResponse();
mech.setCollisionResponse (femA, femB, resp);  // OK
mech.setCollisionResponse (femC, femD, resp);  // ERROR
```

The complete set of methods for managing collision responses are similar to those for behaviors,

```java
getCollisionResponse (collidable0, collidable1)
setCollisionResponse (collidable0, collidable1, response)
setCollisionResponse (collidable0, collidable1)
clearCollisionResponse (collidable0, collidable1)
clearCollisionResponses ()
```

where `getCollisionResponse()` and `clearCollisionResponse()` respectively return and clear response components that have been previously set using one of the `setCollisionResponse()` methods.

7.8.2 Available information

Information that can be obtained from a `CollisionResponse` component includes whether or not the collidable is in collision, the current contact points, forces and pressures acting on the vertices of the colliding meshes, and the
underlying CollisionHandler objects that maintain the contact constraints between each colliding mesh. Much of the available information is similar to that which can be displayed using collision rendering (Section 7.5).

The information methods provided by CollisionResponse are listed below. Many take a cidx argument to request information for either the first or second collidable, which refer, respectively, to the collidables specified by the arguments collidable0 and collidable1 in the setCollisionResponse() methods.

- boolean inContact()
  Queries if the collidables associated with the response are in contact. Note that this may be true even if the method getContactData() returns an empty list; this can occur, for instance, for vertex penetration contact when the collision meshes intersect in a small region without any interpenetrating vertices.

- List<ContactData> getContactData()
  Returns a list of the most recently computed contacts between the collidables. Each contact is described by a ContactData object supplying information about the contact points on each collidable, the normal, and contact and friction forces. If there are no current contacts, the returned list is empty. For more details on this information, consult the API documentation for ContactData and ContactPoint.

- Map<Vertex3d, Vector3d> getContactForces(int cidx)
  Returns a map of the contact forces acting on the vertices of the collision meshes of either the first or second collidable, as specified by setting cidx to 0 or 1. This information is most useful and accurate when using vertex penetration contact (Section 7.4.1.2).

- Map<Vertex3d, Vector3d> getContactPressures(int cidx)
  Returns a map of the contact pressures acting on the vertices of the collision meshes of either the first or second collidable, as specified by setting cidx to 0 or 1. This information is most useful and accurate when using vertex penetration contact (Section 7.4.1.2).

- ArrayList<PenetrationRegion> getPenetrationRegions(int cidx)
  Returns a list of all the penetration regions on either the first or second collidable, as specified by setting cidx to 0 or 1. Penetration regions are available only if the collision manager’s collider type is set to AJL_CONTOUR (Section 7.4.2).

- ArrayList<CollisionHandler> getHandlers()
  Returns the CollisionHandlers for all currently active collisions associated with the collidables of the response.

A typical usage scenario for collision responses is to create them before the simulation is run, possibly in the root model build() method, and then query them when the simulation is running, such from the apply() method of a Monitor (Section 5.3). For example, in the root model build() method, the response could be created with the call

```java
CollisionResponse resp = mech.setCollisionResponse (femA, femB);
```

and then used in some runtime code as follows:

```java
Map<Vertex3d, Vector3d> collMap = resp.getContactForces (0);
```

If for some reason it is difficult to store a reference to the response between its construction and its use, then getCollisionResponse() can be used to retrieve it:

```java
CollisionResponse resp = mech.getCollisionResponse (femA, femB);
Map<Vertex3d, Vector3d> collMap = resp.getContactForces (0);
```

As with contact rendering, the information returned by a collision response may sometimes appear to lag the simulation by one time step. That is because contacts are computed at the end of each time step \( i \), and then used to compute the contact forces during the next step \( i + 1 \). The information returned by a response at the end of step \( i \) is thus based on contacts detected at the end of step \( i - 1 \), along with contact forces that are computed during step \( i \) and used to calculate the updated positions and velocities at the end of that step.
7.8.3 Example: monitoring contact forces

A simple example of using a CollisionResponse is given by

`artisynth.demos.tutorial.ContactForceDemo`

This shows an FEM model of a ball rolling down an inclined plane, with a collision response combined with a Monitor used to print out the contact positions and forces at each time step. Contact forces are also rendered in the viewer. The model’s source code, excluding include statements, is shown below:

```java
public class ContactForceMonitor extends RootModel {

    private class ContactMonitor extends MonitorBase {
        public void apply (double t0, double t1) {
            // get the contacts from the collision response and print their
            // positions and forces.
            List<ContactData> cdata = myResp.getContactData();
            if (cdata.size() > 0) {
                System.out.println {
                    "num contacts: "+ cdata.size() + ", time=", t0);
                for (ContactData cd : cdata) {
                    System.out.print {
                        " pos: " + cd.getPosition0().toString("%8.3f"));
                    System.out.println {
                        ", force: " + cd.getContactForce().toString("%8.1f"));
                }
            }
        }
    }

    public void build (String[] args) {
        MechModel mech = new MechModel ("mech");
        addModel (mech);

        // create an FEM ball model
        FemModel3d ball =
            FemFactory.createIcosahedralSphere {
                null, /*radius=*/0.7, /*ndivisions=*/1, /*quality=*/2);
        ball.setName ("ball");
        ball.setSurfaceRendering (SurfaceRender.Shaded);
    }
```
mech.addModel(ball);

// reposition the ball to an appropriate "drop" position
ball.transformGeometry(new RigidTransform3d(-0.5, 0, 2.5, 0, 0.1, 0.1));

// create an inclined plane for the ball to collide with
RigidBody plane = RigidBody.createBox("plane", 4.0, 2.5, 0.25, /* density=*/1000);
plane.setPose(new RigidTransform3d(0, 0, 0, 0, Math.PI/5, 0));
plane.setDynamic(false);
mech.addRigidBody(plane);

// Enable collisions between the ball and the plane. Specifying the ball
// first means contact forces will be rendered in the direction acting on
// the ball.
mech.setCollisionBehavior(ball, plane, true, 0.3);
myResp = mech.setCollisionResponse(ball, plane);

// add a monitor to print out contact positions and forces
addMonitor(new ContactMonitor());

// Render properties: set the collision manager to render contact and
// friction forces, with a scale factor of 0.0001
CollisionManager cm = mech.getCollisionManager();
cm.setDrawContactForces(true);
cm.setDrawFrictionForces(true);
cm.setContactForceLenScale(0.0001);
RenderProps.setVisible(cm, true);
RenderProps.setSolidArrowLines(cm, 0.02, Color.BLUE);

// Render properties: for the ball, make the elements invisible, and
// render its surface as a wire frame to make it easy to see through
RenderProps.setVisible(ball.getElements(), false);
RenderProps.setLineColor(ball, new Color(.8f, .8f, 1f));
RenderProps.setFaceStyle(ball, FaceStyle.NONE);
RenderProps.setDrawEdges(ball, true);
RenderProps.setEdgeWidth(ball, 2); // wire frame edge width
RenderProps.setFaceColor(plane, new Color(.7f, 1f, .7f));
}

The build method creates a simple FEM ball and inclined plane, and positions the ball to an appropriate drop position (lines 27-42). The collisions are enabled between the ball and the plane, along with a CollisionResponse that is stored in the global reference myResp to allow it to be accessed from the monitor (lines 47-48).

The monitor itself is implemented by the class ContactMonitor, which is created by subclassing MonitorBase and overriding the apply() method to print out the contact information (lines 5-21). It does this by using the response’s getContactData() method to obtain a list of the contacts, and if there are any, printing the number of contacts, the time step start time (\( t_0 \)), and the position and force of each contact. An instance of the monitor is created and added to the root model at line 51.

Rendering is set so that the collision manager renders both the contact and friction forces in the viewer (lines 55-60), using blue arrows with a radius of 0.02 (line 60) and a scale factor for the arrow length of 0.0001 (line 58). To make the ball easy to see through, its elements are made invisible, and instead is rendered using only its surface mesh, with face rendering disabled and edges drawn with a width of 2 (lines 63-67).

To run this example in ArtiSynth, select All demos > tutorial > ContactForceMonitor from the Models menu. Running the model will result in the ball colliding with the plane, showing the contact and friction forces as blue arrows (Figure 7.19), while the monitor prints out the contact positions and forces to the console at each time step, producing output like this:

```
num contacts: 2, time=0.64
pos: -0.918 0.059 0.821, force: 17508.5 0.0 24098.4
pos: -0.760 -0.285 0.706, force: 11607.9 0.0 15976.9
num contacts: 2, time=0.65
pos: -0.918 0.059 0.821, force: 16989.6 0.0 23384.2
pos: -0.760 -0.285 0.706, force: 11955.8 0.0 16455.7
num contacts: 3, time=0.66
```
7.9 Tips and limitations

This section describes practical tips that can be employed when using ArtiSynth’s contact simulation, together with some of its limitations.

7.9.1 Contact jitter

ArtiSynth’s attempt to resolve the interpenetration of colliding bodies may sometimes cause a jittering behavior around the colliding area, as the surface collides, separates, and re-collides. This can usually be stabilized by maintaining a certain interpenetration distance during contact. This distance is controlled by the `MechModel` property `penetrationTol`. ArtiSynth attempts to compute a suitable default value for this property, but for some applications it may be necessary to control the value explicitly using the `MechModel` methods:

```plaintext
setPenetrationTol (double dist)
double getPenetrationTol ()
```

7.9.2 Passing through objects

The ArtiSynth collision detection mechanism is static, which means that mesh intersections are computed at a fixed point in time and do not presently utilize velocity information. This means that if the colliding objects are fast enough or thin enough, it is possible for them to pass completely through each other during a simulation step (Figure 7.20, left). Similarly, even if objects do not pass completely through each other, the penetration may be large enough for contacts to be established with the wrong side (7.20, right).

![Figure 7.20: If object speeds are high enough or the objects thin enough, it is possible for collidables to pass completely through each other (left), or to penetrate sufficiently that contacts are established with the wrong side (right).](image)

When this happens, there are two straightforward solutions:

1. Reduce the simulation step size.
2. Increase the thickness of one or more of the colliding meshes. This option is facilitated by the fact that, as mentioned in Section 7.3, collision meshes can be independent of a collidable’s physical geometry.

The problem of collidables passing through each other can be particularly acute in the case of shell elements, and consequently collisions involving shell elements are not fully supported. However, collisions involving shell elements will work in some circumstances, as described in Section 7.1.4.
7.9.3 Stray vertices

The stray vertex problem sometimes occurs when vertex penetration contact is used with objects with sharp edges. Because vertex penetration contacts are determined by finding the nearest face to each vertex on the opposing mesh, this may sometimes cause an inappropriate face to be selected if penetration is deep enough and near a sharp turn in the opposing mesh (Figure 7.21).

![Image of vertex penetration contact near a sharp turn](image)

Figure 7.21: Vertex penetration contact near a sharp turn in the opposing mesh may sometimes cause the contact to be directed toward the wrong face, as seen here for the upper left vertex of the penetrating square.

Possible solutions include:

1. Reduce the simulation step size.
2. Use the VERTEX_EDGE_PENETRATION contact method (Section 7.4.1.3).
3. Adjust the mesh geometry or increase the mesh resolution; higher mesh resolutions will increase the number of contacts and reduce the impact of stray vertices.

7.9.4 Coulomb friction and stability

ArtiSynth uses a “box” friction approximation [8] to compute Coulomb (dry) friction, which allows for a less expensive and more robust computation at the expense of some accuracy. Box friction computes friction forces in two orthogonal directions in the plane perpendicular to the contact normal, using a fixed normal force taken from the previous solve, instead of employing the more detailed polyhedralized friction cones commonly used in multibody dynamics [1, 16]. The resulting linear complementarity problem is convex and hence more easily solved. Errors can be minimized by ensuring that one of the friction directions is parallel to the tangential contact velocity.

By default, friction forces are computed after after the main velocity solve, in a secondary solve that does not use implicit integration techniques. This reduces compute time and works well for rigid bodies, or for FEM models when the friction forces are relatively small (which they often are in biomechanical applications). However, applications involving FEM models and large friction forces may suffer from instability. An example of this might be an FEM model resting on an inclined plane and relying on friction to remain stationary under a large load. To handle such situations, one can enable implicit friction integration by setting the useImplicitFriction property in MechModel; this can be done in code using the MechModel methods

```java
setUseImplicitFriction (boolean enable)
```

```java
boolean getUseImplicitFriction()
```

At the time of this writing, implicit friction integration is a new ArtiSynth feature, and should be considered to be in beta testing.

When using implicit friction integration, if redundant contacts arise (typically between rigid bodies), it may be necessary to regularize both the contact constraints, using one of the methods described in Section 7.7, as well as the friction constraints, by setting the frictionCreep property of either the collision manager or behavior to a non-zero value, as described in Section 7.2.1.
Chapter 8

Muscle Wrapping and Via Points

ArtiSynth provides support for multipoint springs and muscles, which are similar to axial springs and muscles (Sections 3.1.1 and 4.4), except that they can contain multiple via points and also wrap around obstacles. This allows the associated force directions to vary in response to obstacles and constraints in the model, which is particularly important in biomechanical models where point-to-point muscles need to wrap around anatomical structures such as bones. A schematic illustration is shown in Figure 8.1, where a single spring connects points $p_0$ and $p_2$, while passing through a single via point $p_1$ and wrapping around obstacles $W_1$ and $W_2$. Figure 8.2 shows two examples involving a rigid body with fixed via points and a spring wrapping around three rigid bodies.

Figure 8.1: Schematic illustration of a multipoint spring passing through a via point $p_1$ and wrapping around two obstacles $W_1$ and $W_2$. The points $A_1, B_1$ and $A_2, B_2$ denote the first and last locations where $W_1$ and $W_2$ make contact with the spring.

Figure 8.2: Left: A multipoint spring with two via points rigidly fixed to a box-shaped rigid body. Right: A multipoint spring wrapped around three obstacles.

As with axial springs and muscles, multipoint springs and muscles must have two points to denote their beginning and end. In between, they can have any number of via points, which are fixed locations which the spring must pass through in the specified order. Any ArtiSynth Point object may be specified as a via point, including particles and markers. The purpose of the via point is generally to direct the spring along some particular path. In particular, the path directions
before and after a via point will generally be different, and forces acting on the via point will be determined by the
tension in the spring (or muscle) acting along these two different directions.

![Figure 8.3: A multipoint spring with a single via point p₁, showing the unit direction vectors u_A and u_B immediately before and after.](image)

Conceptually, the spring or muscle “slides” through its via points, which act analogously to virtual three-dimen-
sional pulleys. In particular, the proportional distance between via points does not remain fixed.

The tension $f$ within the spring or muscle is computed from its material, using the relation $f(l, l', a)$ described in
Sections 3.1.1 and 4.4.1, where $l$ now denotes the entire length of the spring as it passes through the via points and
wraps around obstacles. The total force $f$ acting on each via point is then given by

$$f = f(u_B - u_A)$$

where $u_B$ and $u_A$ are unit vectors giving the spring’s direction immediately after and before the via point (Figure 8.3).

Multipoint springs can also be made to wrap around one or more wrappable objects. Unlike via points, wrappable
objects can occur in any order along the spring and wrapping only occurs when the spring and the object actually
collide. Any ArtiSynth object that implements Wrappable can be used as a wrapping object (currently, only RigidBody
objects implement Wrappable). The forces acting on a wrappable are those generated by the forces $f_A$ and $f_B$ acting on
the points A and B where the spring makes and leaves contact with the it (Figure 8.4). These forces are given by

$$f_A = -fu_A, \quad f_B = fu_B,$$

where $u_B$ are $u_A$ are unit vectors giving the spring’s direction immediately before A and after B. Points A and B are
collectively known as the A/B points.

![Figure 8.4: A multipoint spring wrapping around a single obstacle W, with initial and final contact at points A and B, and associated unit direction vectors u_A and u_B.](image)

### 8.1 Via Points

Multipoint springs and muscles are implemented by the classes MultiPointSpring and MultiPointMuscle, respectively.
The relationship between MultiPointSpring and MultiPointMuscle is the same as that between AxialSpring and Muscle:
The latter is a subclass of the former, and allows the creation of active tension forces in response to its excitation
property.
An application allocates one of these components, sets the appropriate material properties for the tension forces, and then adds points and wrappable objects as desired.

Points can be added, queried, and removed using the methods

```java
void addPoint (Point pnt)
Point getPoint (int idx)
int numPoints()
boolean removePoint (Point pnt)
```

As with `AxialSpring`, there must be at least two points anchoring the beginning and end of the spring. Any additional points will be *via points*.

Figure 8.5: A multipoint spring with two segments, separated by a blue via point (top), with the rightmost segment set to be wrappable so that it can wrap around a cylinder. The right image shows the wrappable segment’s knots.

The section of a multipoint spring between any two adjacent points is known as a *segment*. By default, each segment forms a straight line between the two points and does *not* interact with any wrappable obstacles. To interact with wrappables, a segment needs to be declared *wrappable*, as described in Section 8.2.

Spring construction is illustrated by the following code fragment:

```java
MultiPoint spring = new MultiPointSpring();
spring.setMaterial (new LinearAxialMaterial (stiffness, damping));
spring.addPoint (p0); // start point
spring.addPoint (p1); // via point
spring.addPoint (p2); // via point
spring.addPoint (p3); // stop point
```

This creates a new `MultiPointSpring` and sets its material to a simple linear material with a specified stiffness and damping. Four points `p0, p1, p2, p3` are then added, forming a start point, two via points, and a stop point.

### 8.1.1 Example: a muscle with via points

A simple example of a muscle containing via points is given by `artisynth.demos.tutorial.ViaPointMuscle`. It consists of a `MultiPointMuscle` passing through two via points attached to a block. The code is given below:

```java
package artisynth.demos.tutorial;
import java.awt.Color;
import artisynth.core.gui.ControlPanel;
import artisynth.core.materials.SimpleAxialMuscle;
```
public class ViaPointMuscle extends RootModel {

    protected static double size = 1.0;

    public void build (String[] args) {
        MechModel mech = new MechModel("mech");
        addModel (mech);

        mech.setFrameDamping (1.0); // set damping parameters
        mech.setRotaryDamping (0.1);

        // create block to which muscle will be attached
        RigidBody block = RigidBody.createBox ("block", 
                                           "widths="/1.0, 1.0, 1.0,"
                                           "density="/1.0);
        mech.addRigidBody (block);

        // create muscle start and end points
        Particle p0 = new Particle ("mass="/0.1, 
                                    
                                    "x,y,z="/-3.0, 0, 0.5);
        p0.setDynamic (false);
        mech.addParticle (p0);
        Particle p1 = new Particle ("mass="/0.1, 
                                    "x,y,z="/3.0, 0, 0.5);
        p1.setDynamic (false);
        mech.addParticle (p1);

        // create markers to serve as via points
        FrameMarker via0 = new FrameMarker();
        mech.addFrameMarker (via0, block, new Point3d (-0.5, 0, 0.5));
        FrameMarker via1 = new FrameMarker();
        mech.addFrameMarker (via1, block, new Point3d (0.5, 0, 0.5));

        // create muscle, set material, and add points
        MultiPointMuscle muscle = new MultiPointMuscle ();
    }
}
muscle.setMaterial(new SimpleAxialMuscle(/*k=*/1, /*d=*/0, /*maxf=*/10));
muscle.addPoint(p0);
muscle.addPoint(via0);
muscle.addPoint(vial);
muscle.addPoint(p1);
mech.addMultiPointSpring(muscle);

// set render properties
RenderProps.setSphericalPoints(mech, 0.1, Color.WHITE);
RenderProps.setCylindricalLines(mech, 0.03, Color.RED);
createControlPanel();

private void createControlPanel() {
    // creates a panel to adjust the muscle excitation
    ControlPanel panel = new ControlPanel("options", "");
    panel.addWidget(this, "models/mech/multiPointSprings/0: excitation");
    addControlPanel(panel);
}

Lines 21-30 of the build() method create a MechModel and add a simple rigid body block to it. Two non-dynamic points (p0 and p1) are then created to act as muscle end points (lines 33-38), along with two markers (via0 and vial) which are attached to the block to act as via points (lines 41-44). The muscle itself is created by lines 42-53, with the end points and via points being added in order from start to end. The muscle material is a SimpleAxialMuscle, which computes tension according to the simple linear formula (4.1) described in Section 4.4.1. Lines 56-57 set render properties for the model, and line 59 creates a control panel (Section 5.1) that allows the muscle excitation property to be interactively controlled.

To run this example in ArtiSynth, select All demos > tutorial > ViaPointMuscle from the Models menu. The model should load and initially appear as in Figure 8.6. Running the model will cause the block to fall and swing about under gravity, while changing the muscle’s excitation in the control panel will vary its tension.

### 8.2 Obstacle Wrapping

As mentioned in Section 8.1, segments between pairs of via points can be declared wrappable, allowing them to interact with wrappable obstacles. This can be done as via points are added to the spring, using the methods

```java
void setSegmentWrappable(int numKnots)
void setSegmentWrappable(int numKnots, Point3d[] initialPoints)
```

These make wrappable the next segment to be created (i.e., the segment between the most recently added point and the next point to be added), with numKnots specifying the number of knots that should be used to implement the wrapping. Knots are points that divide the wrappable segment into a piecewise linear curve, and are used to check for collisions with the wrapping surfaces (Figure 8.5). The argument initialPoints used by the second method is an optional argument which, if non-null, can be used to specify intermediate guide points to give the segment an initial path around any obstacles (for more details, see Section 8.4).

Each wrappable segment will be capable of colliding with any of the wrappable obstacles that are known to the spring. Wrappables can be added, queried and removed using the following methods:

```java
void addWrappable(Wrappable wrappable)
Wrappable getWrappable(int idx)
int numWrappables()
boolean removeWrappable(Wrappable wrappable)
```

Unlike points, however, there is no implied ordering and wrappables can be added in any order and at any time during the spring’s construction.

Wrappable spring construction is illustrated by the following code fragment:
MultiPoint spring = new MultiPointSpring();
spring.setMaterial(new LinearAxialMaterial(stiffness, damping));
spring.addPoint(p0); // start point
spring.setSegmentWrappable(50); // wrappable segment
spring.addPoint(p1); // via point
spring.addPoint(p2); // end point
spring.addWrappable(wrappable1);
spring.addWrappable(wrappable2);
spring.updateWrapSegments(); // 'shrink wrap' spring to the obstacles

This creates a new MultiPointSpring with a linear material and three points p0, p1, and p2, forming a start point, via point, and stop point. The segment between p0 and p1 is set to be wrappable with 50 knot points. Two wrappable obstacles are added next, each of which will interact with the p0-p1 segment, but not with the non-wrappable p1-p2 segment. Finally, updateWrapSegments() is called to do an initial solve for the wrapping segments, so that they will be "pulled tight" around any obstacles before simulation begins.

It is also possible to make a segment wrappable after spring construction, using the method

void setSegmentWrappable(int segIdx, int numKnots, Point3d[] initialPoints)

where segIdx identifies the segment between points segIdx and segIdx + 1.

How many knots should be specified for a wrappable segment? Enough so that the resulting piecewise-linear approximation to the wrapping curve is sufficiently "smooth", and also enough to adequately detect contact with the obstacles without passing through them. Values between 50 and 100 generally give good results. Obstacles that are small with respect to the segment length may necessitate more knots. Making the number of knots very large will slow down the computation (although the computational cost is only $O(n)$ with respect to the number of knots).

At the time of this writing, ArtiSynth implements two types of Wrappable object, both of which are instances of RigidBody. The first are specialized analytic subclasses of RigidBody, listed in Table 8.1, which define specific geometries and use analytic methods for the collision handling with the knot points. The use of analytic methods allows for greater accuracy and (possibly) computational efficiency, and so because of this, these special geometry wrappables should be used whenever possible.

<table>
<thead>
<tr>
<th>Wrappable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RigidCylinder</td>
<td>A cylinder with a specified height and radius</td>
</tr>
<tr>
<td>RigidSphere</td>
<td>A sphere with a specified radius</td>
</tr>
<tr>
<td>RigidEllipsoid</td>
<td>An ellipsoid with specified semi-axis lengths</td>
</tr>
<tr>
<td>RigidTorus</td>
<td>A torus with specified inner and outer radii</td>
</tr>
</tbody>
</table>

Table 8.1: Specialized analytic subclasses of RigidBody

The second are general rigid bodies which are not analytic subclasses, and for which the wrapping surface is determined directly from the geometry of its collision mesh returned by getCollisionMesh(). (Typically the collision mesh corresponds to the surface mesh, but it is possible to specify alternates; see Section 3.2.9.) This is useful in that it permits wrapping around arbitrary mesh geometries (Figure 8.7), but in order for the wrapping to work well, these geometries should be smooth, without sharp edges or corners. Wrapping around general meshes is implemented using a quadratically interpolated signed-distance grid (Section 4.5), and the resolution of this grid also affects the effective smoothness of the wrapping surface. More details on this are given in Section 8.3.

8.2.1 Example: wrapping around a cylinder

A example showing multipoint spring wrapping is given by artisynth.demos.tutorial.CylinderWrapping. It consists of a MultiPointSpring passing through a single via point, with both segments on either side of the point made wrappable. Two analytic wrappables are used: a fixed RigidCylinder, and a moving RigidEllipsoid attached to the end of the spring. The code, excluding include directives, is given below:

```java
def public class CylinderWrapping extends RootModel {
  public void build (String[] args) {
    MechModel mech = new MechModel ("mech");
  }
```
Figure 8.7: Muscle strands wrapped around general bone-shaped meshes: a humerus (left), and a pelvis (right)

Figure 8.8: CylinderWrapping model loaded into ArtiSynth.

```java
addModel (mech);
mech.setFrameDamping (100.0); // set damping parameters
tsetRotaryDamping (10.0);

double density = 150;

Particle via0 = new Particle (/* mass=*/0, /*x,y,z=*/-1.0, 0.0, 4.0);
via0.setDynamic (false);
mech.addParticle (via0);
Particle p1 = new Particle (/* mass=*/0, /*x,y,z=*/-3.0, 0.0, 0.0);
p1.setDynamic (false);
mech.addParticle (p1);

// create cylindrical wrapping object
RigidCylinder cylinder = new RigidCylinder ("cylinder", /* rad=*/0.5, /* height=*/3.5, density, /* nsides=*/50);
cylinder.setPose (new RigidTransform3d (0, 0, 1.5, 0, 0, Math.PI/2));
cylinder.setDynamic (false);
```
Lines 4-17 of the `build()` method create a MechModel with two fixed particles `via0` and `p1` to be used as via and stop points. Next, two analytic wrappables are created: a RigidCylinder and a RigidEllipsoid, with the former fixed in place and the latter connected to the start of the spring via the marker `p0` (lines 20-37). Collisions are enabled between these two wrappables at line 40. The spring itself is created (lines 44-52), using `setSegmentWrappable()` to make the segments (`p0, via0`) and (`via0, p1`) wrappable with 50 knots each, and `addWrappable()` to make it aware of the two wrappables. Finally, render properties at set (lines 55-58), and a control panel (Section 5.1) is added that allows the spring’s `drawKnots` and `drawABPoints` properties to be interactively set.

To run this example in ArtiSynth, select `All demos > tutorial > CylinderWrapping` from the Models menu. The model should load and initially appear as in Figure 8.8. Running the model will cause the ellipsoid to fall and the spring to wrap around the cylinder. Using the pull tool (Section “Pull Manipulation” in the ArtiSynth User Interface Guide) on the ellipsoid can cause additional motions and make it also collide with the spring. Selecting `drawKnots` or `drawABPoints` in the control panel will cause the spring to render its knots and/or A/B points.
8.3 General Surfaces and Distance Grids

As mentioned in Section 8.2, wrapping around general mesh geometries is implemented using a quadratically interpolated signed distance grid. By default, for a rigid body, this grid is generated automatically from the body’s collision mesh (as returned by `getCollisionMesh()`; see Section 7.3).

Using a distance grid allows very efficient collision handling between the body and the wrap segment knots. However, it also means that the true wrapping surface is not actually the collision mesh itself, but instead the zero-valued isosurface associated with quadratic grid interpolation. Well-behaved wrapping behavior requires that this isosurface be smooth and free of sharp edges, so that knot motions remain relatively smooth as they move across it. Quadratic interpolation helps with this, which is the reason for employing it. Otherwise, one should try to ensure that (a) the collision mesh from which the grid is generated is itself smooth and free of sharp edges, and (b) the grid has sufficient resolution to not introduce discretization artifacts.

Muscle wrapping is often performed around structures such as bones, for which the representing surface mesh is often insufficiently smooth (especially if segmented from medical image data). In some cases, the distance grid’s quadratic interpolation may provide sufficient smoothing on its own; to determine this, one should examine the quadratic isosurface as described below. In other cases, it may be necessary to explicitly smooth the mesh itself, either externally or within ArtiSynth using the `LaplacianSmother` class, which can apply iterations of either Laplacian or volume-preserving Taubin smoothing, via the method

```java
LaplacianSmother.smooth (mesh, numi, lam, mu);
```

Here `numi` is the number of iterations and `tau` and `mu` are the Taubin parameters. Setting `lam` = 1 and `mu` = 0 results in traditional Laplacian smoothing. If this causes the mesh to shrink more than desired, one can counter this by setting `tau` and `mu` to values used for Taubin smoothing, as described in [20].

If the mesh is large (i.e., has many vertices), then smoothing it may take noticeable computational time. In such cases, it is generally best to simply save and reuse the smoothed mesh.

By default, if a rigid body contains only one polygonal mesh, then its surface and collision meshes (returned by `getSurfaceMesh()` and `getCollisionMesh()`), respectively) are the same. However, if it is necessary to significantly smooth or modify the collision mesh, for wrapping or other purposes, it may be desirable to use different meshes for the surface and collision. This can be done by making the surface mesh non-collidable and adding an additional mesh that is collidable, as discussed in Section 3.2.9 as illustrated by the following code fragment:

```java
PolygonalMesh surfaceMesh;
PolygonalMesh wrappingMesh;

// ... initialize surface and wrapping meshes ...

// create the body from the surface mesh
RigidBody body = RigidBody.createFromMesh (  
    "body", mesh, /* density=*/1000, /* scale=*/1.0);

// set the surface mesh to be non-collidable, and add the wrapping mesh as  
// collidable but not having mass
body.getSurfaceMeshComp().setIsCollidable (false);
RigidMeshComp wcomp = body.addMesh (  
    wrappingMesh, /* hasMass=*/false, /*collidable=*/true);
RenderProps.setVisible (wcomp, false); // hide the wrapping mesh
```

Here, to ensure that the wrapping mesh does not contribute to the body’s inertia, its `hasMass` property is set to false.

Although it is possible to specify a collision mesh that is separate from the surface mesh, there is currently no way to specify separate collision meshes for wrapping and collision handling. If this is desired for some reason, then one alternative is to create a separate body for wrapping purposes, and then attach it to the main body, as described in Section 8.5.

To verify that the distance grid’s quadratic isosurface is sufficiently smooth for wrapping purposes, it is useful to visualize the both distance grid and its isosurface directly, and if necessary adjust the resolution used to generate the grid. This can be accomplished using the body’s `DistanceGridComp`, which is a subcomponent named `distanceGrid` and which may be obtained using the method
A `DistanceGridComp` exports a number of properties that can be used to control the grid’s visualization, resolution, and fit around the collision mesh. These properties are described in detail in Section 4.5, and can be set either in code using their set/get accessors, or interactively using custom control panels or by selecting the grid component in the GUI and choosing Edit properties ... from the right-click context menu.

When rendering the mesh isosurface, it is usually desirable to also disable rendering of the collision meshes within the rigid body. For convenience, this can be accomplished by setting the body’s `gridSurfaceRendering` property to `true`, which will cause the grid isosurface to be rendered instead of the body’s meshes. The isosurface type will be that indicated by the grid component’s `surfaceType` property (which should be `QUADRATIC` for the quadratic isosurface), and the rendering will occur independently of the visibility settings for the meshes or the grid component.

### 8.3.1 Example: wrapping around a bone

An example of wrapping around a general mesh is given by `artisynth.demos.tutorial.TalusWrapping`. It consists of a `MultiPointSpring` anchored by two via points and wrapped around a rigid body representing a talus bone. The code, with include directives omitted, is given below:

```java
public class TalusWrapping extends RootModel {

    private static Color BONE = new Color(1f, 1f, 0.8f);
    private static double DTOR = Math.PI/180.0;

    public void build (String[] args) {
        MechModel mech = new MechModel("mech");
        addModel (mech);

        // read in the talus bone mesh
        PolygonalMesh mesh = null;
        try {
            mesh = new PolygonalMesh {
                PathFinder.findSourceDir(this) + "*/data/TalusBone.obj");
            
        }
        catch (Exception e) {
            System.out.println ("Error reading mesh:" + e);
        }
    }
}
```
ArtiSynth Modeling Guide 241

// smooth the mesh using 20 iterations of regular Laplacian smoothing
LaplacianSmoother.smooth (mesh, /*count=*/20, /*lambda=*/1, /*mu=*/0);

// create the talus body from the mesh
RigidBody talus = RigidBody.createFromMesh {
  "talus", mesh, /*density=*/1000, /*scale=*/1.0);
mech.addRigidBody (talus);
talus.setDynamic (false);
RenderProps.setFaceColor (talus, BONE);

// create start and end points for the spring
Particle p0 = new Particle (/*mass=*/0, /*x,y,z=*/2, 0, 0);
p0.setDynamic (false);
mech.addParticle (p0);
Particle p1 = new Particle (/*mass=*/0, /*x,y,z=*/-2, 0, 0);
p1.setDynamic (false);
mech.addParticle (p1);

// create a wrappable spring using a SimpleAxialMuscle material
MultiPointSpring spring = new MultiPointSpring ("spring");
spring.setMaterial (new SimpleAxialMuscle (/*k=*/0.5, /*d=*/0, /*maxf=*/0.04));

spring.setSegmentWrappable (100, new Point3d[] { new Point3d (0.0, -1.0, 0.0) });

spring.updateWrapSegments (); // update the wrapping path
mech.addMultiPointSpring (spring);

// set render properties
DistanceGridComp gcomp = talus.getDistanceGridComp ();
RenderProps.setSphericalPoints (mech, 0.05, Color.BLUE); // points
RenderProps.setLineWidth (gcomp, 0); // normal rendering off
RenderProps.setCylindricalLines (spring, 0.03, Color.RED); // spring
RenderProps.setSphericalPoints (spring, 0.05, Color.WHITE); // knots

// create a control panel for interactive control
ControlPanel panel = new ControlPanel ();
panel.addWidget (talus, "gridSurfaceRendering");
panel.addWidget (gcomp, "resolution");
panel.addWidget (gcomp, "maxResolution");
panel.addWidget (gcomp, "renderGrid");
panel.addWidget (gcomp, "rendererRanges");
panel.addWidget (spring, "drawKnots");
panel.addWidget (spring, "wrapDamping");
addControlPanel (panel);

The mesh describing the talus bone is loaded from the file "data/TalusBone.obj" located beneath the model’s source directory (lines 11-19), with the utility class PathFinder used to determine the file path (Section 2.6). To ensure better wrapping behavior, the mesh is smoothed using Laplacian smoothing (line 21) before being used to create the rigid body (lines 23-27). The spring and its anchor points p0 and p1 are created between lines 30-49, with the talus added as a wrappable. The spring contains a single segment which is made wrappable using 100 knots, and initialized with an intermediate point (line 45) to ensure that it wraps around the bone in the correct way. Intermediate points are described in more detail in Section 8.4.

Render properties are set at lines 52-56; this includes turning off rendering for grid normals by zeroing the lineWidth render property for the grid component.

Finally, lines 59-67 create a control panel (Section 5.1) for interactively controlling a variety of properties, including gridSurfaceRendering for the talus (to see the grid isosurface instead of the bone mesh), resolution, maxResolution,
renderGrid, and renderRanges for the grid component (to control its resolution and visibility), and drawKnots and wrapDamping for the spring (to make knots visible and to adjust the wrap damping as described in Section 8.6).

To run this example in ArtiSynth, select All demos > tutorial > TalusWrapping from the Models menu. Since all of the dynamic components are fixed, running the model will not cause any initial motion. However, while simulating, one can use the viewer’s graphical dragger fixtures (see the section “Transformer Tools” in the ArtiSynth User Interface Guide) to move p0 or p1 and hence pull the spring across the bone surface (Figure 8.9, left). One can also interactively adjust the property settings in the control panel to view the grid, isosurface, and knots, and the adjust the grid’s resolution. Figure 8.9, right, shows the model with renderGrid and drawKnots set to true and renderRanges set to "10:12 * *".

8.4 Initializing the Wrap Path

By default, when a multipoint spring or muscle is initialized (either at the start of the simulation or as a result of calling updateWrapSegments()), each wrappable segment is initialized to a straight line between its via points. This path is then adjusted to avoid and wrap around obstacles, using artificial linear forces as described in Section 8.6. The result is a local shortest path that wraps around obstacles instead of penetrating them. However, in some cases, the initial path may not be the one desired; instead, one may want it to wrap around obstacles some other way. This can be achieved by specifying additional intermediate points to initialize the segment as a piecewise linear path which threads its way around obstacles in the desired manner (Figure 8.10). These are specified using the optional initialPnts argument to the setSegmentWrappable() methods.

Figure 8.10: By default, the path for each wrappable segment is initialized to a straight line between its via points (dotted line, left), which is then adjusted to wrap around obstacles (solid line, middle). To cause the path to wrap around obstacles in a different way, it can instead be initialized using a piecewise-linear path defined by intermediate initial points (dotted line, right), which will then adjust to an alternate configuration.

When initial points are specified, it is recommended to finish construction of the spring or muscle with a call to updateWrapSegments(). This fits the wrappable segments to their correct path around the obstacles, which can then be seen immediately when the model is first loaded. On the other hand, by omitting an initial call to updateWrapSegments(), it is possible to see the initial path as specified by the initial points. This may be useful to verify that they are in the correct locations.

In some cases, initial points may also be necessary to help ensure that the initial path does not penetrate obstacles. While obstacle penetration will normally be resolved by the artificial forces described in Section 8.6, this may not always work correctly if the starting path penetrates an obstacle too deeply.

8.4.1 Example: wrapping around a torus

An example of using initial points is given by artisynth.demos.tutorial.TorusWrapping, in which a spring is wrapped completely around the inner section of a torus. The primary code for the build method is given below:

```
MechModel mech = new MechModel ("mech");
addModel (mech);
mech setFrameDamping (1.0); // set damping parameters
mech setRotaryDamping (10.0);

// create the torus
double DTOR = Math.PI/180;
```
The mech model is created in the usual way with frame and rotary damping set to 1 and 10 (lines 4-5). The torus is created using the analytic wrappable RigidTorus (lines 8-14). The spring start and end points p0 and p1 are created at lines (17-22), and the spring itself is created at lines (26-41), with six initial points being specified to setSegmentWrappable() to wrap the spring completely around the torus inner section.

To run this example in ArtiSynth, select All demos > tutorial > TorusWrapping from the Models menu. The torus will slide along the wrapped spring until it reaches equilibrium.
8.5 Alternate Wrapping Surfaces

Although it common to use the general mesh geometry of a RigidBody as the wrapping surface, situations may arise where it is desirable to not do this. These may include:

- The general mesh geometry is not sufficiently smooth to form a good wrapping surface;
- Wrapping around the default mesh geometry is not stable, in that it is too easy for the wrap strand to “slip off”;
- Using one of the simpler analytic geometries (Table 8.1) may result in a more efficient computation.

There are a couple of ways to handle this. One, discussed in Section 8.3, involves creating a collision mesh which is separate from the general mesh geometry. However, that same collision mesh must then also be used for collision handling (Chapter 7). If that is undesirable, or if multiple wrapping surfaces are needed, then a different approach may be used. This involves creating the desired wrappable as a separate object and then attaching it to the main RigidBody. Typically, this wrappable will be created with zero mass (or density), so that it does not alter the effective mass or inertia of the main body. The general procedure then becomes:

1. Create the main RigidBody with whatever desired geometry and inertia is needed;
2. Create the additional wrappable object(s), usually with zero density/mass;
3. Attach the wrapables to the main body using one of the MechModel attachFrame() methods described in Section 3.6.3.

8.5.1 Example: wrapping for a finger joint

An example using an alternate wrapping surface is given by artisynth.demos.tutorial.PhalanxWrapping, which shows a muscle wrapping around a joint between two finger bones. Because the bones themselves are fairly narrow, using them as wrapping surfaces would likely lead to the muscle slipping off. Instead, a RigidCylinder is used for the wrapping and attached to one of the bones. The code, with include directives excluded, is given below:

```java
public class PhalanxWrapping extends RootModel {
    private static Color BONE = new Color (1.0f, 1.0f, 0.8f);
    private static double DTOR = Math.PI/180.0;
    private Rigidbody createBody (MechModel mech, String name, String fileName) {
```
// creates a bone from its mesh and adds it to a MechModel
String filePath = PathFinder.findSourceDir(this) + "/data/" + fileName;
RigidBody body = RigidBody.createFromMesh {
  name, filePath, /* density=*/1000, /* scale=*/1.0);
mech.addRigidBody (body);
RenderProps.setFaceColor (body, BONE);
return body;
}

public void build (String[] args) {
  MechModel mech = new MechModel ("mech");
  addModel (mech);

  // create the two phalanx bones, and offset them
  RigidBody proximal = createBody (mech, "proximal", "HP3ProximalLeft.obj");
  RigidBody distal = createBody (mech, "distal", "HP3MiddleLeft.obj");
  distal.setPose (new RigidTransform3d (0.02500, 0.00094, -0.03979));

  // make the proximal phalanx non dynamic; add damping to the distal
  proximal.setDynamic (false);
  distal.setFrameDamping (0.03);

  // create a revolute joint between the bones
  RigidTransform3d TJW =
    new RigidTransform3d (0.018, 0, -0.022, 0, 0, -DTOR*90);
  HingeJoint joint = new HingeJoint (proximal, distal, TJW);
  joint.setShaftLength (0.02); // render joint as a blue cylinder
  RenderProps.setFaceColor (joint, Color.BLUE);
  mech.addBodyConnector (joint);

  // create markers for muscle origin and insertion points
  FrameMarker origin = mech.addFrameMarkerWorld (proximal, new Point3d (0.0098, -0.0001, -0.0037));
  FrameMarker insertion = mech.addFrameMarkerWorld (distal, new Point3d (0.0293, 0.0009, -0.0415));

  // create a massless RigidCylinder to use as a wrapping surface and
  // attach it to the distal bone
  RigidCylinder cylinder = new RigidCylinder ("wrapSurface", /* rad=*/0.005, /* h=*/0.04, /* density=*/0, /* nsegs=*/32);
  cylinder.setPose (TJW);
  mech.addRigidBody (cylinder);
  mech.attachFrame (cylinder, distal);

  // create a wrappable muscle using a SimpleAxialMuscle material
  MultiPointSpring muscle = new MultiPointMuscle ("muscle");
  muscle.setMaterial (new SimpleAxialMuscle (/* k=*/0.05, /* d=*/0, /* maxf=*/0.04));
  muscle.addPoint (origin);
  // add an initial point to the wrappable segment to make sure it wraps
  // around the cylinder the right way
  muscle.setSegmentWrappable (50, new Point3d [] { new Point3d (0.025, 0.0, -0.02) });
  muscle.addPoint (insertion);
  muscle.addWrappable (cylinder);
  muscle.updateWrapSegments (); // "'shrink wrap'" around cylinder
  mech.addMultiPointSpring (muscle);

  // set render properties
  RenderProps.setSphericalPoints (mech, 0.002, Color.BLUE);
  RenderProps.setCylindricalLines (muscle, 0.001, Color.RED);
  RenderProps.setFaceColor (cylinder, new Color (200, 200, 230));
}
The method `createBody()` (lines 6-14) creates a rigid body from a geometry mesh stored in a file in the directory “data” beneath the source directory, using the utility class `PathFinder` used to determine the file path (Section 2.6).

Within the `build()` method, a `MechModel` is created containing two rigid bodies representing the bones, `proximal` and `distal`, with `proximal` fixed and `distal` free to move with a frame damping of 0.03 (lines 18-28). A cylindrical joint is then added between the bones, along with markers describing the muscle’s origin and insertion points (lines 31-42). A `RigidCylinder` is created to act as a wrapping obstacle and attached to the `distal` bone in the same location as the joint (lines 46-50); since it is created with a density of 0 it has no mass and hence does not affect the bone’s inertia. The muscle itself is created at lines 53-64, using a `SimpleAxialMuscle` as a material and an extra initial point specified to `setSegmentWrappable()` to ensure that it wraps around the cylinder in the correct way (Section 8.4). Finally, some render properties are set at lines 67-69.

To run this example in ArtiSynth, select `All demos > tutorial > PhalanxWrapping` from the Models menu. The model should load and initially appear as in Figure 8.12. When running the model, one can move the `distal` bone either by using the pull tool (Section “Pull Manipulation” in the `ArtiSynth User Interface Guide`), or selecting the muscle in the GUI, invoking a property dialog by choosing `Edit properties` ... from the right-click context menu, and adjusting the excitation property.

### 8.6 Tuning the Wrapping Behavior

Wrappable segments are implemented internally using artificial linear elastic forces to draw the knots together and keep them from penetrating obstacles. These artificial forces are invisible to the simulation: the wrapping segment has no mass, and the knot forces are used to create what is essentially a first order physics that “shrink wraps” each segment around the obstacles at the beginning of each simulation step, forming a shortest-distance geodesic curve from which the wrapping contact points A and B are calculated. This process is now described in more detail.

Assume that a wrappable segment has \( m \) knots, indexed by \( k = 1, \ldots, m \), each located at a position \( x_k \). Two types of artificial forces then act on each knot: a wrapping force that pulls it closer to other knots, and contact forces that push it away from wrappable obstacles. The wrapping force is given by

\[
 f_{w,k} = K_w (x_{k+1} - 2x_k + x_{k-1})
\]

where \( K_w \) is the wrapping stiffness. To determine the contact forces, we compute, for each wrappable, the knot’s distance to the surface \( d_k \) and associated normal direction \( n_k \), where \( d_k < 0 \) implies that the knot is inside. These quantities are determined either analytically (for analytic wrappables, Table 8.1), or using a signed distance grid (for general wrappables, Section 8.3). The contact forces are then given by

\[
 f_{c,k} = \begin{cases} 
 -K_c d_k n_k & \text{if } d_k < 0 \\
 0 & \text{otherwise,}
\end{cases}
\]

where \( K_c \) is the contact stiffness.

The total force \( f_k \) acting on each knot is then given by

\[
 f_k = f_{w,k} + \sum_{c} f_{c,k}
\]

where the latter term is the sum of contact forces for all wrappables. If we let \( x \) and \( f \) denote the aggregate position and force vectors for all knots, then computing the wrap path involves finding the equilibrium position such that \( f(x) = 0 \). This is done at the beginning of each simulation step, or whenever `updateWrapSegments()` is called, and is achieved iteratively using Newton’s method. If \( x' \) and \( f(x') \) denote the positions and forces at iteration \( j \), and

\[
 K = \frac{\partial f}{\partial x}
\]

denotes the local force derivative (or “stiffness”), then the basic Newton update is given by

\[
 x^{j+1} = x^j - K^{-1} f(x^j).
\]

In practice, to help deal with the nonlinearities associated with contact, we use a damped Newton update,

\[
 x^{j+1} = x^j + \alpha (I - K)^{-1} f(x^j),
\]  

(8.1)
where $D$ is a constant wrap damping parameter, and $\alpha$ is an adaptively computed step size adjustment. The computation of (8.1) can be performed quickly, in $O(m)$ time, since $K$ is a block-tridiagonal matrix, and the number of iterations required is typically small (on the order of 10 or less), particularly since the iterative procedure continues across simulation steps and so $f(x)$ does not need to be brought to 0 for any given step. The maximum number of Newton iterations used for each time step is $N_{\text{max}}$.

Again, it is important to understand the artificial knot forces $f(x)$ described here are separate from the physical spring/muscle tension forces $f(l, \dot{l}, a)$ discussed in Sections 3.1.1 and 4.4.1, and only facilitate the computation of each wrappable segment’s path around obstacles.

The default values for the wrapping parameters are $K_w = 1$, $K_c = 10$, $D = 10$, and $N_{\text{max}} = 10$, and these often give satisfactory results without the need for modification. However, in some situations the default muscle wrapping may not perform adequately and it is necessary to adjust these parameters. Problems may include:

- The wrapping path does not settle down and tends to “jump around”. Solutions include increasing the damping parameter $D$ or the maximum number of wrap iterations $N_i$. For general wrapping surfaces (Section 8.3), one should also ensure that the surface is sufficiently smooth.
- A wrapping surface is too thin and so the wrapping path “jumps through” it. Solutions include increasing the damping parameter $D$, increasing the number of knots in the segment, or decreasing the simulation step size. An alternative approach is to use an alternative wrapping surface (Section 8.5) that is thicker and better behaved.

Wrapping parameters are exported as properties of MultiPointSpring and MultiPointMuscle, and may be changed in code (using their set/get accessors), or interactively, either by exposing them through a control panel, or by selecting the spring/muscle in the GUI and choosing Edit properties ... from the right-click context menu. Property values include:

- **wrapStiffness** Wrapping stiffness $K_w$ between knot points (default value 1). Since the wrapping behavior is determined by the damping to stiffness ratio, it is generally not necessary to change this value.
- **wrapDamping** Damping factor $D$ (default value 10). Increasing this value relative to $K_w$ results in wrap path motions that are smoother and less likely to penetrate obstacles, but which are also less dynamically responsive. Applications generally work with damping values between 10 and 100 (assuming $K_w = 1$).
- **contactStiffness** Contact stiffness $K_c$ used to resolve obstacle penetration (default value 10). It is generally not necessary to change this value. Decreasing it will increase the distance that knots are permitted to penetrate obstacles, which may result in a slightly more stable contact behavior.
- **maxWrapIterations** Maximum number of Newton iterations $N_{\text{max}}$ per time step (default value 10). If the wrapping simulation exhibits instability, particularly with regard to obstacle contact, increasing the number of iterations (to say 100) may help.

In addition, MultiPointSpring and MultiPointMuscle also export the following properties to control the rendering of knot and A/B points:

- **drawKnots** If true, renders the knot points in each wrappable segment. This can be useful to visualize the knot density. Knots are rendered using the style, size, and color given by the pointStyle, pointRadius, pointSize, and pointColor values of the spring/muscle’s render properties.
- **drawABPoints** If true, renders the A/B points. These are the first and last points of contact that a wrap segment makes with each wrappable, and correspond to the points where the spring/muscle’s tension acts on that wrappable (Section 8 and Figure 8.4). A/B points are rendered using the style and size given by the pointStyle, pointRadius ($\times 1.2$) and pointSize values of the spring/muscle’s render properties, and the color given by the ABPointColor property.
Chapter 9

Skinning

A useful technique for creating anatomical and biomechanical models is to attach a passive mesh to an underlying set of dynamically active bodies so that it deforms in accordance with the motion of those bodies. ArtiSynth allows meshes to be attached, or “skinned”, to collections of both rigid bodies and FEM models, facilitating the creation of structures that are either embedded in, or connect or envelope a set of underlying components. Such approaches are well known in the computer animation community, where they are widely used to represent the deformable tissue surrounding a “skeleton” of articulated rigid bodies, and have more recently been applied to biomechanics [13].

Figure 9.1: A skin mesh used to delimit the boundary of the human upper airway, connected to various surrounding structures including the palate, tongue, and jaw [19].

One application of skinning is to create a continuous skin surrounding an underlying set of anatomical components. For example, for modeling the human airway, a disparate set of models describing the tongue, jaw, palate and pharynx can be connected together with a surface skin to form a seamless airtight mesh (Figure 9.1), as described in [19]. This then provides a uniform boundary for handling air or fluid interactions associated with tasks such as speech or swallowing.

ArtiSynth provides support for “skinning” a mesh over an underlying set of master bodies, consisting of rigid bodies and/or FEM models, such that the mesh vertices deform in response to changes in the position, orientation and shape of the master bodies.

9.1 Implementation

This section describes the technical details of the ArtiSynth skinning mechanism. A skin mesh is implemented using a SkinMeshBody, which contains a base mesh and references to a set of underlying dynamic master bodies. A master body can be either a Frame (of which RigidBody is a subclass), or a FemModel3d. The positions of the mesh vertices
(along with markers and other points that can be attached to the skin mesh) are determined by a weighted sum of influences from each of the \( m \) master bodies, such that as the latter move and/or deform, the vertices and attached points deform as well. More precisely, for each master body \( k, k \in \{0, \ldots, m - 1\} \), let \( w_k \) be the weighting factor and \( f_k(\mathbf{q}_k) \) the connection function that describes the contribution of body \( k \) to the position of the vertices (or attached points) as a function of its generalized coordinates \( \mathbf{q}_k \). Then if the position of a vertex (or attached point) is denoted by \( \mathbf{p} \) and its initial (or base) position is \( \mathbf{p}_0 \), we have

\[
\mathbf{p} = \sum_{k=0}^{m-1} w_k f_k(\mathbf{q}_k) + w_m \mathbf{p}_0. \tag{9.1}
\]

The weight \( w_m \) in the last term is known as the base weight and describes an optional contribution from the base position \( \mathbf{p}_0 \). Usually \( w_m = 0 \), unless the vertex is not connected to any master bodies, in which case \( w_m = 1 \), so that the vertex is anchored to its initial position.

In general, connection weights \( w_k \) are computed based on the distances \( d_k \) between the vertex (or attached point) and each master body \( k \). More details on this are given in Sections 9.2 and 9.3.

For Frame master bodies, the connection function is one associated with various rigid body skinning techniques known in the literature. These include linear, linear dual quaternion, and iterative dual quaternion skinning. Which technique is implemented is determined by the frameBlending property of the SkinMeshBody, which can be queried or set in code using the methods

\[
\text{FrameBlending} \quad \text{getFrameBlending}() \\
\text{void setFrameBlending(} \text{FrameBlending} \text{ blending)}
\]

where FrameBlending is an enumerated type defined by SkinMeshBody with the following values:

**LINEAR**

Linear blending, in which the connection function \( f_k() \) implements a standard rigid connection between the vertex and the frame coordinates. Let the frame’s generalized coordinates \( \mathbf{q}_k \) be given by the \( 3 \times 3 \) rotation matrix \( \mathbf{R} \) and translation vector \( \mathbf{p}_F \) describing its pose, with its initial pose given by \( \mathbf{R}_0 \) and \( \mathbf{p}_{F0} \). The connection function \( f_k() \) then takes the form

\[
f_k(\mathbf{R}, \mathbf{p}_F) = \mathbf{R} \hat{\mathbf{q}}_k(\mathbf{p}_0 - \mathbf{p}_{F0}) + \mathbf{p}_F. \tag{9.2}
\]

Linear blending is faster than other blending techniques but is more prone to pinching and creasing artifacts in the presence of large rotations between frames.

**DUAL_QUATERNION_LINEAR**

Linear dual quaternion blending, which is more computationally expensive but typically gives better results than linear blending, and is described in detail as DLB in [6]. Let the frame’s generalized coordinates \( \mathbf{q}_k \) be given by the dual-quaternion \( \hat{\mathbf{q}}_k \) (describing both rotation and translation), with the initial pose given by the dual-quaternion \( \hat{\mathbf{q}}_{00} \). Then define the relative dual-quatervnion \( \hat{\mathbf{q}}_k \) as

\[
\hat{\mathbf{q}}_k = \frac{\hat{\mathbf{q}}_k \hat{\mathbf{q}}_{00}^{-1}}{\| \sum_j w_j \hat{\mathbf{q}}_j \hat{\mathbf{q}}_{j0}^{-1} \|}, \tag{9.3}
\]

where the denominator is formed by summing over all master bodies \( j \) which are frames. The connection function \( f_k() \) is then given by

\[
f_k(\hat{\mathbf{q}}_k) = \hat{\mathbf{q}}_k \hat{\mathbf{q}}_k^{-1} \hat{\mathbf{q}}_{0} - \hat{\mathbf{q}}_{0}, \tag{9.4}
\]

where we note that a dual quaternion multiplied by a position vector yields a position vector.

**DUAL_QUATERNION_ITERATIVE**

Dual quaternion iterative blending, which is a more complex dual quaternion technique described in detail as DIB in [6]. The connection function for iterative dual quaternion blending involves an iterative process and is not described here. It also does not conform to (9.1), because the connection functions \( f_k() \) for the Frame master bodies do not combine linearly. Instead, if there are \( r \) Frame master bodies, there is a single connection function

\[
f(w_0, \ldots, w_{r-1}, \hat{\mathbf{q}}_0, \ldots, \hat{\mathbf{q}}_{r-1}) \tag{9.5}
\]

that determines the connection for all of them, given their weighting factors \( w_j \) and generalized coordinates \( \hat{\mathbf{q}}_j \).

Iterative blending relies on two parameters: a blend tolerance, and a maximum number of blend steps, both of which are controlled by the SkinMeshBody properties DQBlendTolerance and DQMaxBlendSteps, which have default values of \( 1^{-8} \) and 3.
Iterative dual quaternion blending is not completely supported in ArtiSynth. In particular, because of its complexity, the associated force and velocity mappings are computed using the simpler computations employed for linear dual quaternion blending. For the examples shown in this chapter, iterative dual quaternion gives results that are quite close to those of linear dual quaternion blending.

For FEM master bodies, the connection works by tying each vertex (or attached point) to a specific FEM element using a fixed-length offset vector \( \mathbf{d} \) that rotates in conjunction with the element. This is illustrated in Figure 9.2 for the case of a single FEM master body. Starting with the initial vertex position \( \mathbf{p}_0 \), we find the nearest point \( \mathbf{p}_{e0} \) on the nearest FEM element, along with the offset vector \( \mathbf{d}_0 = \mathbf{p}_0 - \mathbf{p}_{e0} \). The point \( \mathbf{p}_{e0} \) can be expressed as the weighted sum of the initial element nodal positions \( \mathbf{x}_j \),

\[
\mathbf{p}_{e0} = \sum_{j=0}^{n-1} \alpha_j \mathbf{x}_j, \tag{9.6}
\]

where \( n \) is the number of nodes and \( \alpha_j \) represent the (constant) nodal coordinates. As the element moves and deforms, the element point \( \mathbf{p}_e \) moves with the nodal positions \( \mathbf{x}_j \) according to the same relationship, while the offset vector \( \mathbf{d} \) rotates according to \( \mathbf{d} = \mathbf{R}_E \mathbf{d}_0 \), where \( \mathbf{R}_E \) is the rotation of the element’s coordinate frame \( E \) with respect to its initial orientation. The connection function \( f_k() \) then takes the form

\[
f_k(\mathbf{x}_0, \ldots, \mathbf{x}_{n-1}) = \sum_{j=0}^{n-1} \alpha_j \mathbf{x}_j + \mathbf{R}_E \mathbf{d}_0. \tag{9.7}
\]

\( \mathbf{R}_E \) is determined by computing a polar decomposition \( \mathbf{F} = \mathbf{R}_E \mathbf{P} \) on the deformation gradient \( \mathbf{F} \) at the element’s center. We note that the displacement \( \mathbf{d} \) is only rotated and so the distance \( \|\mathbf{d}\| = \|\mathbf{d}_0\| \) of the vertex from the element remains constant. If the vertex is initially on or inside the element, then \( \mathbf{d}_0 = \mathbf{0} \) and (9.7) takes the form of a standard point/element attachment as described in 6.4.3.

Figure 9.2: Illustration of FEM skinning, showing how a position \( \mathbf{p} \) is tied to an FEM element. Given the initial position \( \mathbf{p}_0 \), we find the nearest point \( \mathbf{p}_{e0} \) on the element, along with the offset vector \( \mathbf{d}_0 = \mathbf{p}_0 - \mathbf{p}_{e0} \) (left). As the element moves and deforms, the updated position is obtained from \( \mathbf{p} = \mathbf{p}_e + \mathbf{d} \), where \( \mathbf{p}_e \) deforms with the element, and \( \mathbf{d} \) rotates in tandem with its coordinate frame \( E \).

While it is sometimes possible to determine weights \( \alpha_j \) that control a vertex position outside an element, without the need for an offset vector \( \mathbf{d} \), the resulting vertex positions tend to be very sensitive to element distortions, particularly when the vertex is located at some distance. Keeping the element-vertex distance constant via an offset vector usually results in more plausible skinning behavior.

### 9.2 Creating a skin mesh

As mentioned above, skin meshes within ArtiSynth are implemented using the `SkinMeshBody` component. Applications typically create a skin mesh in code according to the following steps:
1. Create an instance of `SkinMeshBody` and assign its underlying mesh, usually within the constructor;
2. Add references to the required master bodies;
3. Compute the master body connections

This is illustrated by the following example:

```java
MechModel mech;
PolygonalMesh mesh;

// ... initialize mesh ...

// create the body with an underlying mesh:
SkinMeshBody skinMesh = new SkinMeshBody(mesh);

// add references to the master bodies:
skinMesh.addMasterBody(rigidBody1);
skinMesh.addMasterBody(rigidBody2);
skinMesh.addMasterBody(femModel1);

// compute the weighted connections for each vertex:
skinMesh.computeAllVertexConnections();

// add to the MechModel
mech.addMeshBody(skinMesh)
```

Master body references are added using `addMasterBody()`. When all the master bodies have been added, the method `computeAllVertexConnections()` computes the weighted connections to each vertex. The connection weights $w_k$ for each vertex are determined by a **weighting function**, based on the distances $d_k$ between the vertex and each master body. The default weighting function is **inverse-square weighting**, which first computes a set of raw weights $w_k^*$ according to

$$
w_k^* = \frac{d_{min}^2}{d_k^2}, \quad (9.8)
$$

where $d_{min} \equiv \min(d_j)$ is the minimum master body distance, and then normalizes these to determine $w_k$:

$$
w_k = \frac{w_k^*}{\sum_j w_j^*}. \quad (9.9)
$$

Other weighting functions can be specified, as described in Section 9.3.

`SkinMeshBody` provides the following set of methods to set and query its master body configuration:

```java
void addMasterBody (ModelComponent body) // add a master body
int numMasterBodies () // query number of master bodies
boolean hasMasterBody (ModelComponent body) // query master body presence
ModelComponent getMasterBody (int idx) // get a master body by index
RigidTransform3d getBasePose (Frame frame) // query base pose for frame
void setBasePose (Frame frame, RigidTransform3d T) // set base pose for frame
```

When a `Frame` master body is added using `addMasterBody()`, its initial, or base, pose (corresponding to $R_0$ and $p_{F0}$ in Section 9.1) is set from its current pose. If necessary, applications can later query and reset the base pose using the methods `getBasePose()` and `setBasePose()`.

Internally, each vertex is connected to the master bodies by a `PointSkinAttachment`, which contains a list of `PointSkinAttachment.SkinConnection` components describing each master connection. Applications can obtain the `PointSkinAttachment` for each vertex using the method

```java
PointSkinAttachment getVertexAttachment (int vidx)
```

where `vidx` is the vertex index, which must be in the range 0 to `numv-1`, with `numv` the number of vertices as returned by `numVertices()`.

Methods also exist to query and set each vertex’s base (i.e., initial) position $p_0$:

```java
Point3d getVertexBasePosition (int vidx)
void setVertexBasePosition (int vidx, Point3d pos)
```
Figure 9.3: RigidBodySkinning model loaded into ArtiSynth (left), then run with excitation set to 0.7 (middle). The right image shows the result of changing the frameBlending property from LINEAR to DUAL_QUATERNION_LINEAR.

9.2.1 Example: skinning over rigid bodies

An example of skinning a mesh over two rigid bodies is given by artisynth.demos.tutorial.RigidBodySkinning. It consists of a SkinMeshBody placed around two rigid bodies connected by a hinge joint to form a toy “arm”, with a Muscle added to move the lower body with respect to the upper. The code for the build() method is given below:

```java
public void build (String[] args) throws IOException {
    MechModel mech = new MechModel("mech");
    addModel (mech);

    // set damping parameters for rigid bodies
    mech setFrameDamping (10);
    mech.setRotaryDamping (100.0);

    // create a toy "arm" consisting of upper and lower rigid bodies connected
    // by a revolute joint:
    double len = 2.0;
    RigidBody upper = addBody (mech, "upper");
    upper.setPose (new RigidTransform3d (0, 0, len/2));
    upper.setDynamic (false); // upper body is fixed

    RigidBody lower = addBody (mech, "lower");
    // reposition the lower body
    double angle = Math.toRadians (225);
    double sin = Math.sin (angle);
    double cos = Math.cos (angle);
    lower.setPose (new RigidTransform3d (sin*len/2, 0, cos*len/2, 0, angle, 0));

    // add the revolute joint between the upper and lower bodies:
    HingeJoint joint =
        new HingeJoint (lower, upper, new Point3d(), Vector3d.Y_UNIT);
    joint.setName ("elbow");
    mech.addBodyConnector (joint);

    // add two frame markers and a "muscle" to move the lower body
    FrameMarker mku = mech.addFrameMarker (upper, new Point3d (-len/20, 0, len/2.4));
    FrameMarker mkl = mech.addFrameMarker (lower, new Point3d (len/20, 0, -len/4));
    Muscle muscle = new Muscle("muscle");
```
```java
muscle.setMaterial(new SimpleAxialMuscle(1000.0, 0, 2000.0));
mech.attachAxialSpring(mku, mk1, muscle);

// create an ellipsoidal base mesh for the SkinMeshBody by scaling a spherical mesh
PolygonalMesh mesh = MeshFactory.createSphere(1.0, 12, 12);
mesh.scale(1, 1, 2.5);
new RigidTransform3d(-0.6, 0, 0, Math.toRadians(22.5), 0);

// create the skinMesh, with the upper and lower bodies as master bodies
SkinMeshBody skinMesh = new SkinMeshBody(mesh);
skinMesh.addMasterBody(upper);
skinMesh.addMasterBody(lower);
skinMesh.computeAllVertexConnections();
mech.addMeshBody(skinMesh);

// add a control panel to adjust the muscle excitation and frameBlending
ControlPanel panel = new ControlPanel();
panel.addWidget(muscle, "excitation");
panel.addWidget(skinMesh, "frameBlending");
addControlPanel(panel);

// set up render properties
RenderProps.setFaceStyle(skinMesh, Renderer.FaceStyle.NONE);
RenderProps.setDrawEdges(skinMesh, true);
RenderProps.setLineColor(skinMesh, Color.CYAN);
RenderProps.setSpindleLines(muscle, 0.06, Color.RED);
RenderProps.setSphericalPoints(mech, 0.05, Color.WHITE);
RenderProps.setFaceColor(joint, Color.BLUE);
joint.setShaftLength(len/3);
joint.setShaftRadius(0.05);
RenderProps.setFaceColor(mech, new Color(0.8f, 0.8f, 0.8f));
```

A MechModel is created in the usual way (lines 2-7). To this is added a very simple toy “arm” consisting of an upper and lower body connected by a hinge joint (lines 9-27), with a simple point-to-point muscle attached between frame markers on the upper and lower bodies to provide a means of moving the arm (lines 29-36). Creation of the arm bodies uses an addBody() method which is not shown.

The mesh to be skinned is an ellipsoid, created using the FemFactory method createSphere() to produce a spherical mesh which is then scaled and repositioned (lines 38-43). The skin body itself is then created around this mesh, with the upper and lower bodies assigned as master bodies and the connections computed using computeAllVertexConnections() (lines 45-50).

A control panel is added to allow control over the muscle’s excitation as well as the skin body’s frameBlending property (lines 52-56). Finally, render properties are set (lines 58-67): the skin mesh is made transparent by setting its faceStyle and drawEdges properties to NONE and true, respectively, with cyan colored edges; the muscle is rendered as a red spindle; the joint is drawn as a blue cylinder and the bodies are colored light grey.

To run this example in ArtiSynth, select All demos > tutorial > RigidBodySkinning from the Models menu. The model should load and initially appear as in Figure 9.3 (left). When running the simulation, the arm can be flexed by adjusting the muscle excitation property in the control panel, causing the skin mesh to deform (Figure 9.3, middle). Changing the frameBlending property from its default value of LINEAR to DUAL_QUATERNION_LINEAR causes the mesh deformation to become fatter and less prone to creasing (Figure 9.3, right).

### 9.3 Computing weights

As described above, the default method for computing skin connection weights is inverse-square weighting (equations 9.8 and 9.9). However, applications can specify alternatives to this. The method

```java
void setGaussianWeighting(double sigma)
```
causes weights to be computed according to a Gaussian weighting scheme, with \( \sigma \) specifying the standard deviation \( \sigma \). Raw weights \( w^*_k \) are then computed according to

\[
 w_i = \exp \left( -\frac{(d_k - d_{\text{min}})^2}{2\sigma^2} \right),
\]

and then normalized to form \( w_k \).

The method

```java
void setInverseSquareWeighting()
```

reverts the weighting function back to inverse-square weighting.

It is also possible to specify a custom weighting function by implementing a subclass of `SkinWeightingFunction`. Subclasses must implement the function

```java
void computeWeights ( double[] weights, Point3d pos, NearestPoint[] nearestPnts);
```

in which the weights for each master body are computed and returned in `weights`. `pos` gives the initial position of the vertex (or attached point) being skinning, while `nearestPnts` provides information about the distance from `pos` to each of the master bodies, using an array of `SkinMeshBody.NearestPoint` objects:

```java
class NearestPoint {
  public Point3d nearPoint; // nearest point on the body
  public double distance; // distance to the body
  public ModelComponent body; // master body (either Frame or FemModel3d)
}
```

Once an instance of `SkinWeightingFunction` has been created, it can be set as the skin mesh weighting function by calling

```java
void setWeightingFunction (SkinWeightingFunction fxn)
```

Subsequent calls to `computeAllVertexConnections()`, or the `addMarker` or `computeAttachment` methods described in Section 9.4, will then employ the specified weighting.

As an example, imagine an application wishes to compute weights according to an inverse-cubic weighting function, such that to

\[
 w_k^* = \frac{d_{\text{min}}^3}{d_k^3}.
\]

A subclass of `SkinWeightingFunction` implementing this could then be defined as

```java
class MyWeighting extends SkinWeightingFunction {
  // implements inverse-cubic weighting
  public void computeWeights ( double[] weights, Point3d pos, NearestPoint[] nearestPnts) {
    // find minimum distance to all the master bodies
    double dmin = Double.POSITIVE_INFINITY;
    for (int i=0; i<nearestPnts.length; i++) {
      if (nearestPnts[i].distance < dmin) {
        dmin = nearestPnts[i].distance;
      }
    }
    double sumw = 0; // sum of all weights (for normalizing)
    // compute raw weights:
    for (int i=0; i<nearestPnts.length; i++) {
      double d = nearestPnts[i].distance;
      double w;
      if (d == dmin) {
        w = 1; // handles case where dmin = d = 0
      } else {
        w = dmin / d;
      }
      weights[i] = w;
    }
  }
}
```
else {
    w = dmin * dmin * dmin / (d * d * d);
} 
weights[i] = w;
sumw += w;
}
// normalize the weights:
for (int i = 0; i < nearestPnts.length; i++) {
    weights[i] /= sumw;
}
}

and then set as the weighting function using the code fragment:

SkinMeshBody skinMesh;
// ...
skinMesh.setWeightingFunction (new MyWeighting());

The current weighting function for a skin mesh can be queried using

SkinWeightingFunction getWeightingFunction()

The inverse-square and Gaussian weighting methods described above are implemented using the system-provided SkinWeightingFunction subclasses InverseSquareWeighting and GaussianWeighting, respectively.

### 9.3.1 Setting weights explicitly

As an alternative to the weighting function, applications can also create connections to vertices or points in which the weights are explicitly specified. This allows for situations in which a weighting function is unable to properly specify all the weights correctly.

When a mesh is initially added to a skin body, via either the constructor SkinMeshBody(mesh), or by a call to setMesh(mesh), all master body connections are cleared and the vertex position is “fixed” to its initial position, also known as its base position. After the master bodies have been added, vertex connections can be created by calling computeAllVertexConnections(), as described above. However, connections can also be created on a per-vertex basis, using the method

```c
void computeVertexConnections (int vidx, VectorNd weights)
```

where `vidx` is the index of the desired vertex and `weights` is an optional argument which if non-null explicitly specifies the connection weights. A sketch example of how this can be used is given in the following code fragment:

```c
VectorNd weights = new VectorNd (skinMesh.numMasterBodies());
// compute connections for each vertex
for (int i = 0; i < skinMesh.numVertices(); i++) {
    // ... compute connections weights as required ...
    skinMesh.computeVertexConnections (i, weights);
}
```

For comparison, it should be noted that the code fragment

```c
for (int i = 0; i < skinMesh.numVertices(); i++) {
    skinMesh.computeVertexConnections (i, null);
}
```

in which weights are *not* explicitly specified, is is equivalent to calling computeAllVertexConnections().

If necessary, after vertex connections have been computed, they can also be cleared, using the method

```c
void clearVertexConnections (int vidx)
```
This will disconnect the vertex with index `vidx` from the master bodies, and set its base weighting \( w_m \) (equation 9.1) to 1, so that it will remain fixed to its initial position.

In some special cases, it may be desirable for an application to set attachment base weights to some value other than 0 when connections are present. Base weights for vertex attachments can be queried and set using the methods

```java
double getVertexBaseWeight (int vidx)
void setVertexBaseWeight (int vidx, double weight, boolean normalize)
```

In the second method, the argument `normalize`, if `true`, causes the weights of the other connections to be scaled so that the total weight sum remains the same. For skin markers and point attachments (Section 9.4), base weights can be set by calling the equivalent `PointSkinAttachment` methods

```java
double getBaseWeight ()
void setBaseWeight (double weight, boolean normalize)
```

(If needed, the attachment for a skin marker can be obtained by calling its `getAttachment()` method.) In addition, base weights can also be specified in the `weights` argument to the method `computeVertexConnections(vidx,weights)`, as well as the methods `addMarker(name,pos,weights)` and `createPointAttachment(pnt,weights)` described in Section 9.4. This is done by giving `weights` a size equal to \( m + 1 \), where \( m \) is the number of master bodies, and specifying the base weight in the last location.

### 9.4 Markers and point attachments

In addition to controlling the positions of mesh vertices, a `SkinMeshBody` can also be used to control the positions of dynamic point components, including markers and other points which can be attached to the skin body. For both markers and attached points, any applied forces are propagated back onto the skin body’s master bodies, using the principle of virtual work. This allows skin bodies to be fully incorporated into a dynamic model.

Markers and point attachments can be created even if the `SkinMeshBody` does not have a mesh, a fact that can be used in situations where a mesh is unnecessary, such as when employing skinning techniques for muscle wrapping (Section 9.7).

#### 9.4.1 Markers

Markers attached to a skin body are instances of `SkinMarker`, and are contained in the body’s subcomponent list `markers` (analogous to the `markers` list for FEM models). Markers can be created and maintained using the following `SkinMeshBody` methods:

```java
// create markers:
SkinMarker addMarker (Point3d pos)
SkinMarker addMarker (String name, Point3d pos)
SkinMarker addMarker (String name, Point3d pos, VectorNd weights)

// remove markers:
boolean removeMarker (SkinMarker mkr)
void clearMarkers ()

// access the marker list:
PointList <SkinMarker> markers()
```

The `addMarker` methods each create and return a `SkinMarker` which is added to the skin body’s list of markers. The marker’s initial position is specified by `pos`, while the second and third methods also allow a name to be specified. Connections between the marker and the master bodies are created in the same way as for mesh vertices, with the connection weights either being determined by the skin body’s weighting function (as returned by `getWeightingFunction()`), or explicitly specified by the argument `weights` (third method).

Once created, markers can be removed individually or all together by the `removeMarker()` and `clearMarkers()` methods. The entire marker list can be accessed on a read-only basis by the method `markers()`.
9.4.2 Point attachments

In addition to markers, applications can also attach any regular Point component (including particles and FEM nodes) to a skin body by using one of its createPointAttachment methods:

```java
PointSkinAttachment createPointAttachment (Point pnt)
PointSkinAttachment createPointAttachment (Point pnt, VectorNd weights)
```

Both of these create a PointSkinAttachment that connects the point pnt to the master bodies in the same way as for mesh vertices and markers, with the connection weights either being determined by the skin body’s weighting function or explicitly specified by the argument weights in the second method.

Once created, the point attachment must also be added to the underlying MechModel, as illustrated by the following code fragment:

```java
MechModel mech;
SkinMeshBody skinBody;
Point pnt;
// ... initialize ...
PointSkinAttachment a = skinBody.createPointAttachment (pnt);
mech.addAttachment (a);
```

9.4.3 Example: skinning rigid bodies and FEM models

An example of skinning a mesh over both rigid bodies and FEM models is given by the demo model artisynth.demos.tutorial.AllBodySkinning. It consists of a skin mesh placed around a tubular FEM model connected to rigid bodies connected at each end, and a marker attached to the mesh tip. The code for the build() method is given below:

```java
public void build (String[] args) { 
    MechModel mech = new MechModel (*mech*);
    addModel (mech);
    // size and density parameters
    double len = 1.0;
    double rad = 0.15;
    double density = 1000.0;
    // create a tubular FEM model, and rotate it so it lies along the x axis
```
A MechModel is first created and length and density parameters are defined (lines 2-8). Then a tubular FEM model is created, by using the `FemFactory` method `createHexTube()` and transforming the result by rotating it by 90 degrees about the y axis (lines 10-14). Two rigid body blocks are then created (lines 16-26) and attached to the ends of the FEM model by finding and attaching the left and rightmost nodes (lines 28-34). The model is anchored to ground by setting the left block to be non-dynamic (line 21).

The mesh to be skinned is a rounded cylinder, created using the `MeshFactory` method `createRoundedCylinder()` and rotating the result by 90 degrees about the y axis (lines 39-44). This is then used to create the skin body itself, to which both rigid bodies and the FEM model are added as master bodies and the vertex connections are computed using a
call to computeAllVertexConnections() (lines 46-52). A marker is then added to the tip of the skin body, using the addMarker() method (lines 54-56). Finally render properties are set (lines 58-64): The mesh is made transparent by drawing only its edges in cyan; the FEM model surface mesh is rendered in blue-grey, and the tip marker is drawn as a red sphere.

To run this example in ArtiSynth, select All demos > tutorial > AllBodySkinning from the Models menu. The model should load and initially appear as in Figure 9.4 (left). When running the simulation, the FEM and the rightmost rigid body fall under gravity, causing the skin mesh to deform. The pull tool can then be used to move things around by applying forces to the master bodies or the skin mesh itself.

### 9.4.4 Mesh-based markers and attachments

For the markers and point attachments described above, the connections to the underlying master bodies are created in the same manner as connections for individual mesh vertices. This means that the resulting markers and attached points move independently of the mesh vertices, as though they were vertices in their own right.

An advantage to this is that such markers and attachments can be created even if the SkinMeshBody does not even have a mesh, as noted above. However, a disadvantage is that such markers will not remain tightly connected to vertex-based features (such as the faces of a PolygonalMesh or the line segments of a PolylineMesh). For example, consider a marker defined by

```java
SkinMarker mkr = skinBody.addMarker (pos);
```

where `pos` is a point that is initially located on a face of the body’s mesh. As the master bodies move and the mesh deforms, the resulting marker may not remain strictly on the face. In many cases, this may not be problematic or the deviation may be too small to matter. However, if it is desirable for markers or point attachments to be tightly bound to mesh features, they can instead be created with the following methods:

```java
// create mesh-based markers:
SkinMarker addMeshMarker (Point3d pos)
SkinMarker addMeshMarker (String name, Point3d pos)

// create mesh-based attachments:
PointSkinAttachment createPointMeshAttachment (Point pnt)
```

The requested position `pos` will then be projected onto the nearest mesh feature (e.g., a face for a PolygonalMesh or a line segment for a PolylineMesh), and the resulting position `p` will be defined as a linear combination of the vertex positions `p_i` for this feature,

\[
p = \sum \beta_i p_i,
\]

where \( \beta_i \) are the barycentric coordinates of `p` with respect to the feature. The master body connections are then defined by the same linear combination of the connections for each vertex. When the master bodies move, the marker or attached point will move with the feature and remain in the same relative position.

Since SkinMeshBody implements the interface PointAttachable, it provides the general point attachment method

```java
PointSkinAttachment createPointAttachment (Point pnt)
```

which allows it to be acted on by agents such as the ArtiSynth marker tool (see the section “Marker tool” in the ArtiSynth User Interface Guide). Whether or not the resulting attachment is a regular attachment or mesh-based is controlled by the skin body’s attachPointsToMesh property, which can be adjusted in the GUI or in code using the property’s accessor methods:

```java
boolean getAttachPointsToMesh ()
void setAttachPointsToMesh (boolean enable)
```

### 9.5 Resolution and Limitations

Skinning techniques do have limitations, which are common to all methodologies.
The passive nature of the connection between skinned vertices and the master bodies means that it can be easy for the skin mesh to self intersect and/or fold and crease in ways that are not physically realistic. These effects are often more pronounced when mesh vertices are relatively far away from the master bodies, or in places where the mesh undergoes a concave deformation. When the master bodies include frames, these effects can sometimes be reduced by setting the `frameBlending` property to `DUAL_QUATERNION_LINEAR` instead of the default `LINEAR`.

When the master bodies include FEM models which undergo large deformations, crimping artifacts may arise if the skin mesh has a higher resolution that the FEM model (Figure 9.5). This is because each mesh vertex is connected to the coordinate frame of a single FEM element, and for reasons of computational efficiency the influence of these coordinate frames is not blended as it is for frame-based master bodies. If crimping artifacts occur, one solution may be to adjust the mesh and/or the FEM model so that their resolutions are more compatible (Figure 9.5, right).

### 9.6 Collisions

It is possible to make skin bodies collide with other ArtiSynth bodies, such as `RigidBody` and `FemModel3d`, which implement the `Collidable` interface (Chapter 7). SkinMeshBody itself implements `CollidableBody`, and is considered a deformable body, so that collisions can be activated either by setting one of the default collision behaviors involving deformable bodies, or by setting an explicit collision behavior between it and another body. Self collisions involving SkinMeshBody are not currently supported.

As described in Section 7.4, collisions work by computing the intersection between the meshes of the skin body and other collidables. The vertices, faces, and (possibly) edges of the resulting intersection region are then used to compute contact constraints, which propagate the effect of the contact back onto the collidable bodies’ dynamic components. For a skin mesh, the dynamic components are the Frame master bodies and the nodes of the FEM master bodies.

Collisions involving SkinMeshBody frequently suffer from the problem of being overconstrained (Section 7.6), whereby the number of contacts exceeds the number of master body DOFs available to handle the collision. This may occur if the skin body contains only rigid bodies, or if the mesh resolution exceeds the resolution of the FEM master bodies. Managing overconstrained collisions is discussed in Section 7.6, with the easiest method being constraint reduction, which can be activated by setting to `true` the `reduceConstraints` property for either the collision manager or a specific `CollisionBehavior` involving the skin body.

Caveats: The distance between the vertices of a skinned mesh and its master bodies can sometimes cause odd or counter-intuitive collision behavior. Collision handling may also be noticeably slower if `frameBlending` is set to `DUAL_QUATERNION_LINEAR` or `DUAL_QUATERNION_ITERATIVE`. If any of the master bodies are FEM models, collisions resulting in large friction forces may result in unstable behavior.
9.6 Example: collision with a cylinder

Collisions involving SkinMeshBody are illustrated by the demo model artisynth.demos.tutorial.SkinBodyCollide, which extends the demo AllBodySkinning to add a cylinder with which the skin body can collide. The code for the demo is given below:

```java
package artisynth.demos.tutorial;

import artisynth.core.femmodels.SkinMeshBody;
import artisynth.core.mechmodels.CollisionBehavior;
import artisynth.core.mechmodels.MechModel;
import artisynth.core.mechmodels.RigidBody;
import maspack.matrix.RigidTransform3d;

public class SkinBodyCollide extends AllBodySkinning {
    public void build (String[] args) {
        super.build (args);

        // get components from the super class
        MechModel mech = (MechModel)models().get("mech");
        SkinMeshBody skinBody = (SkinMeshBody)mech.meshBodies().get("skin");
        RigidBody block0 = mech.rigidBodies().get("block0");

        // set block0 dynamic so the skin body and its masters can
        // fall under gravity
        block0.setDynamic (true);

        // create a cylinder for the skin body to collide with
        RigidBody cylinder =
            RigidBody.createCylinder ("cylinder", 0.5, 2.0, /*density=*/1000.0, /*nsides=*/50);
        cylinder.setDynamic (false);
        cylinder.setPose (new RigidTransform3d (-0.5, 0, -1.5, 0, 0, Math.PI/2));
        mech.addRigidBody (cylinder);

        // enable collisions between the cylinder and the skin body
        CollisionBehavior cb = new CollisionBehavior (true, 0);
        mech.setCollisionBehavior (cylinder, skinBody, cb);
        mech.getCollisionManager().setReduceConstraints (true);
    }
}
```

Figure 9.6: SkinBodyCollide demo, showing a skin mesh colliding with a cylinder about 0.6 seconds into the simulation.
The model subclasses `AllBodySkinning`, using the superclass `build()` method within its own `build` method to create the original model (line 12). It then obtains references to the `MechModel`, `SkinMeshBody`, and leftmost block, using their names to find them in various component lists (lines 14-17). (If the original model had stored references to these components as accessible member attributes, this step would not be needed.)

These component references are then used to make changes to the model: the left block is made dynamic so that the skin mesh can fall freely (line 21), a cylinder is created and added (lines 23-30), collisions are enabled between the skin body and the cylinder (lines 32-34), and the collision manager is asked to use constraint reduction to minimize the chance of overconstrained contact (line 35).

To run this example in ArtiSynth, select All demos > tutorial > SkinBodyCollide from the Models menu. When run, the skin body should fall and collide with the cylinder as shown in Figure 9.6.

### 9.7 Application to muscle wrapping

It is sometimes possible to use skinning as a computationally cheaper way to implement muscle wrapping (Chapter 8). Typically, the end points (i.e., origin and insertion points) of a point-to-point muscle are attached to different bodies. As these bodies move with respect to each other, the path of the muscle may wrap around portions of these bodies and perhaps other intermediate bodies as well. The wrapping mechanism of Chapter 8 manages this by performing the computations necessary to allow one or more wrappable segments of a `MultiPointSpring` to wrap around a prescribed set of rigid bodies. However, if the induced path deformation is not too great, it may be possible to achieve a similar effect at much lower computational cost by simply “skinning” the via points of the multipoint spring to the underlying rigid bodies.

The general approach involves:

1. Creating a `SkinMeshBody` which references as master bodies the bodies containing the origin and insertion points, and possibly other bodies as well;
2. Creating the wrapped muscle using a `MultiPointSpring` with via points that are attached to the skin body.

It should be noted that for this application, the skin body does not need to contain a mesh. Instead, “skinning” connections can be made solely between the master bodies and the via points. An easy way to do this is to simply use skin body markers as via points. Another way is to create the via points as separate particles, and then attach them to the skin body using one of its `createPointAttachment` methods.

> It should also be noted that unlike with the wrapping methods of Chapter 8, skin-based wrapping can be applied around FEM models as well as rigid bodies.

> Generally, we observe better wrapping behavior if the `frameBlending` property of the `SkinMeshBody` is set to `DUAL_QUATERNION_LINEAR` instead of the default value of `LINEAR`.

#### 9.7.1 Example: wrapping for a finger joint

An example of skinning-based muscle wrapping is given by `artisynth.demos.tutorial.PhalanxSkinWrapping`, which is identical to the demo `artisynth.demos.tutorial.PhalanxWrapping` except for using skinning to achieve the wrapping effect. The portion of the code which differs is shown below:

```java
// create a SkinMeshBody and use it to create "skinned" muscle via points
SkinMeshBody skinBody = new SkinMeshBody();
skinBody.addMasterBody (proximal);
skinBody.addMasterBody (distal);
skinBody.setFrameBlending (FrameBlending.DUAL_QUATERNION_LINEAR);
mech.addMeshBody (skinBody);
SkinMarker vial = skinBody.addMarker (new Point3d (0.0215, 0, -0.015));
```
First, a SkinMeshBody is created referencing the proximal and distal bones as master bodies, with the frameBlending property set to \texttt{DUAL QUATERNION LINEAR} (lines 1-6). Next, we create a set of three via points that will be attached to the skin body to guide the muscle around the joint in lieu of making it actually wrap around a cylinder (lines 7-9).
In the original PhalanxWrapping demo, a RigidCylinder was used as a muscle wrap surface. In this demo, we replace this with a simple cylindrical mesh which has no dynamic function but allows us to visualize the wrapping behavior of the via points (lines 11-18). The muscle itself is created using the three via points but with no wrapable segments or bodies (lines 20-30).

A control panel is added to allow for the adjustment of the skin body’s frameBlending property (lines 32-35). Finally, render properties are set as for the original demo, only with the skin body markers rendered as white spheres to make them more visible (lines 37-41).

To run this example in ArtiSynth, select All demos > tutorial > PhalanxSkinWrapping from the Models menu. The model should load and initially appear as in Figure 9.7 (left). The pull tool can then be used to move the distal joint while simulating, to illustrate how well the via points “wrap” around the joint, using the cylindrical mesh as a visual reference (Figure 9.7, middle). Changing the frameBlending property to LINEAR results in a less satisfactory behavior (Figure 9.7, right).
Chapter 10

DICOM Images

Some models are derived from image data, and it may be useful to show the model and image in the same space. For this purpose, a DICOM image widget has been designed, capable of displaying 3D DICOM volumes as a set of three perpendicular planes. An example widget and its property panel is shown in Figure 10.1.

![Figure 10.1: DICOM image of the heart, downloaded from http://www.osirix-viewer.com.](http://www.osirix-viewer.com)

The main classes related to the reading and displaying of DICOM images are:

- **DicomElement**: Describes a single attribute in a DICOM file.
- **DicomHeader**: Contains all header attributes (all but the image data) extracted from a DICOM file.
- **DicomPixelBuffer**: Contains the *decoded* image pixels for a single image frame.
- **DicomSlice**: Contains both the header and image information for a single 2D DICOM slice.
- **DicomImage**: Container for DICOM slices, creating a 3D volume (or 3D + time)
- **DicomReader**: Parses DICOM files and folders, appending information to a DicomImage.
- **DicomViewer**: Displays the DicomImage in the viewer.

If the purpose is simply to display a DICOM volume in the ArtiSynth viewer, then only the last three classes will be of interest. Readers who simply want to display a DICOM image in their model can skip to Section 10.3.
10.1 The DICOM file format

For a complete description of the DICOM format, see the specification page at

http://medical.nema.org/standard.html

which provides a brief description. Another excellent resource is the blog by Roni Zaharia:

http://dicomiseasy.blogspot.ca/

Each DICOM file contains a number of concatenated attributes (a.k.a. elements), one of which defines the embedded binary image pixel data. The other attributes act as meta-data, which can contain identity information of the subject, equipment settings when the image was acquired, spatial and temporal properties of the acquisition, voxel spacings, etc.... The image data typically represents one or more 2D images, concatenated, representing slices (or ‘frames’) of a 3D volume whose locations are described by 13.5 the meta-data. This image data can be a set of raw pixel values, or can be encoded using almost any image-encoding scheme (e.g. JPEG, TIFF, PNG). For medical applications, the image data is typically either raw or compressed using a lossless encoding technique. Complete DICOM acquisitions are typically separated into multiple files, each defining one or few frames. The frames can then be assembled into 3D image ‘stacks’ based on the meta-information, and converted into a form appropriate for display.

<table>
<thead>
<tr>
<th>VR</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CS</td>
<td>Code String</td>
</tr>
<tr>
<td>DS</td>
<td>Decimal String</td>
</tr>
<tr>
<td>DT</td>
<td>Date Time</td>
</tr>
<tr>
<td>IS</td>
<td>Integer String</td>
</tr>
<tr>
<td>OB</td>
<td>Other Byte String</td>
</tr>
<tr>
<td>OF</td>
<td>Other Float String</td>
</tr>
<tr>
<td>OW</td>
<td>Other Word String</td>
</tr>
<tr>
<td>SH</td>
<td>Short String</td>
</tr>
<tr>
<td>UI</td>
<td>Unique Identifier</td>
</tr>
<tr>
<td>US</td>
<td>Unsigned Short</td>
</tr>
<tr>
<td>OX</td>
<td>One of OB, OW, OF</td>
</tr>
</tbody>
</table>

Each DICOM attribute is composed of:

- a standardized unique integer tag in the format (XXXX,XXXX) that defines the group and element of the attribute
- a value representation (VR) that describes the data type and format of the attribute’s value (see Table 10.1)
- a value length that defines the length in bytes of the attribute’s value to follow
- a value field that contains the attribute’s value

This layout is depicted in Figure 10.2. A list of important attributes are provided in Table 10.2.

<table>
<thead>
<tr>
<th>Tag</th>
<th>VR</th>
<th>Value Length</th>
<th>Value Field</th>
</tr>
</thead>
</table>

Figure 10.2: DICOM attribute structure

10.2 The DICOM classes

Each DicomElement represents a single attribute contained in a DICOM file. The DicomHeader contains the collection of DicomElements defined in a file, apart from the pixel data. The image pixels are decoded and stored in a DicomPixelBuffer. Each DicomSlice contains a DicomHeader, as well as the decoded DicomPixelBuffer for a single slice (or ‘frame’). All slices are assembled into a single DicomImage, which can be used to extract 3D voxels and spatial locations from the set of slices. These five classes are described in further detail in the following sections.
Table 10.2: A selection of useful DICOM attributes

<table>
<thead>
<tr>
<th>Attribute name</th>
<th>VR</th>
<th>Tag</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transfer syntax UID</td>
<td>UI</td>
<td>0x0002, 0x0010</td>
</tr>
<tr>
<td>Slice thickness</td>
<td>DS</td>
<td>0x0018, 0x0050</td>
</tr>
<tr>
<td>Spacing between slices</td>
<td>DS</td>
<td>0x0018, 0x0088</td>
</tr>
<tr>
<td>Study ID</td>
<td>SH</td>
<td>0x0020, 0x0010</td>
</tr>
<tr>
<td>Series number</td>
<td>IS</td>
<td>0x0020, 0x0011</td>
</tr>
<tr>
<td>Aquisition number</td>
<td>IS</td>
<td>0x0020, 0x0012</td>
</tr>
<tr>
<td>Image number</td>
<td>IS</td>
<td>0x0020, 0x0013</td>
</tr>
<tr>
<td>Image position patient</td>
<td>DS</td>
<td>0x0020, 0x0032</td>
</tr>
<tr>
<td>Image orientation patient</td>
<td>DS</td>
<td>0x0020, 0x0037</td>
</tr>
<tr>
<td>Temporal position identifier</td>
<td>IS</td>
<td>0x0020, 0x0100</td>
</tr>
<tr>
<td>Number of temporal positions</td>
<td>IS</td>
<td>0x0020, 0x0105</td>
</tr>
<tr>
<td>Slice location</td>
<td>DS</td>
<td>0x0020, 0x1041</td>
</tr>
<tr>
<td>Samples per pixel</td>
<td>US</td>
<td>0x0028, 0x0002</td>
</tr>
<tr>
<td>Photometric interpretation</td>
<td>CS</td>
<td>0x0028, 0x0004</td>
</tr>
<tr>
<td>Planar configuration (color)</td>
<td>US</td>
<td>0x0028, 0x0006</td>
</tr>
<tr>
<td>Number of frames</td>
<td>IS</td>
<td>0x0028, 0x0008</td>
</tr>
<tr>
<td>Rows</td>
<td>US</td>
<td>0x0028, 0x0010</td>
</tr>
<tr>
<td>Columns</td>
<td>US</td>
<td>0x0028, 0x0011</td>
</tr>
<tr>
<td>Pixel spacing</td>
<td>DS</td>
<td>0x0028, 0x0030</td>
</tr>
<tr>
<td>Bits allocated</td>
<td>US</td>
<td>0x0028, 0x0100</td>
</tr>
<tr>
<td>Bits stored</td>
<td>US</td>
<td>0x0028, 0x0101</td>
</tr>
<tr>
<td>High bit</td>
<td>US</td>
<td>0x0028, 0x0102</td>
</tr>
<tr>
<td>Pixel representation</td>
<td>US</td>
<td>0x0028, 0x0103</td>
</tr>
<tr>
<td>Pixel data</td>
<td>OX</td>
<td>0x7FE0, 0x0010</td>
</tr>
</tbody>
</table>

10.2.1 DicomElement

The DicomElement class is a simple container for DICOM attribute information. It has three main properties:

- an integer tag
- a value representation (VR)
- a value

These properties can be obtained using the corresponding get function: getTag(), getVR(), getValue(). The tag refers to the concatenated group/element tag. For example, the transfer syntax UID which corresponds to group 0x0002 and element 0x0010 has a numeric tag of 0x00020010. The VR is represented by an enumerated type, DicomElement.VR. The ‘value’ is the raw value extracted from the DICOM file. In most cases, this will be a String. For raw numeric values (i.e. stored in the DICOM file in binary form) such as the unsigned short (US), the ‘value’ property is exactly the numeric value.

For VRs such as the integer string (IS) or decimal string (DS), the string will still need to be parsed in order to extract the appropriate sequence of numeric values. There are static utility functions for handling this within DicomElement. For a ‘best-guess’ of the desired parsed value based on the VR, one can use the method getParsedValue(). Often, however, the desired value is also context-dependent, so the user should know a priori what type of value(s) to expect. Parsing can also be done automatically by querying for values directly through the DicomHeader object.

10.2.2 DicomHeader

When a DICOM file is parsed, all meta-data (attributes apart from the actual pixel data) is assembled into a DicomHeader object. This essentially acts as a map that can be queried for attributes using one of the following methods:

```java
DicomElement getElement(int tag); // includes VR and data
String getStringValue(int tag); // all non-numeric VRs
String[] getMultiStringValue(int tag); // UT, SH
int getIntValue(int tag, int defaultValue); // IS, DS, SL, UL, SS, US
int[] getMultiIntValue(int tag); // IS, DS, SL, UL, SS, US
```
The first method returns the full element as described in the previous section. The remaining methods are used for convenience when the desired value type is known for the given tag. These methods automatically parse or convert the DicomElement’s value to the desired form.

If the tag does not exist in the header, then the `getIntValue(...)` and `getDecimalValue(...)` will return the supplied `defaultValue`. All other methods will return `null`.

### 10.2.3 DicomPixelBuffer

The `DicomPixelBuffer` contains the decoded image information of an image slice. There are three possible pixel types currently supported:

- byte grayscale values (`PixelType.BYTE`)
- short grayscale values (`PixelType.SHORT`)
- byte RGB values, with layout `RGBRGB...RGB` (`PixelType.BYTE_RGB`)

The pixel buffer stores all pixels in one of these types. The pixels can be queried for directly using `getPixel(idx)` to get a single pixel, or `getBuffer()` to get the entire pixel buffer. Alternatively, a `DicomPixelInterpolator` object can be passed in to convert between pixel types via one of the following methods:

```java
public int getPixelsByte(int x, int dx, int nx, byte[] pixels, int offset, DicomPixelInterpolator interp);
public int getPixelsShort(int x, int dx, int nx, short[] pixels, int offset, DicomPixelInterpolator interp);
public int getPixelsRGB(int x, int dx, int nx, byte[] pixels, int offset, DicomPixelInterpolator interp);
public int getPixels(int x, int dx, int nx, DicomPixelBuffer pixels, int offset, DicomPixelInterpolator interp);
```

These methods populate an output array or buffer with converted pixel values, which can later be passed to a renderer. For further details on these methods, refer to the Javadoc documentation.

### 10.2.4 DicomSlice

A single DICOM file contains both header information, and one or more image ‘frames’ (slices). In ArtiSynth, we separate each frame and attach them to the corresponding header information in a `DicomSlice`. Thus, each slice contains a single `DicomHeader` and `DicomPixelBuffer`. These can be obtained using the methods: `getHeader()` and `getPixelBuffer()`.

For convenience, the `DicomSlice` also has all the same methods for extracting and converting between pixel types as the `DicomPixelBuffer`.

### 10.2.5 DicomImage

An complete DICOM acquisition typically consists of multiple slices forming a 3D image stack, and potentially contains multiple 3D stacks to form a dynamic 3D+time image. The collection of `DicomSlices` are thus assembled into a `DicomImage`, which keeps track of the spatial and temporal positions.

The `DicomImage` is the main object to query for pixels in 3D(+time). To access pixels, it has the following methods:
public int getPixelsByte (  
    int x, int y, int z, int dx, int dy, int dz, int nx, int ny, int nz,  
    int time, byte[] pixels, DicomPixelInterpolator interp);

public int getPixelsShort (  
    int x, int y, int z, int dx, int dy, int dz, int nx, int ny, int nz,  
    int time, short[] pixels, DicomPixelInterpolator interp);

public int getPixelsRGB (  
    int x, int y, int z, int dx, int dy, int dz, int nx, int ny, int nz,  
    int time, byte[] pixels, DicomPixelInterpolator interp);

public int getPixels (  
    int x, int y, int z, int dx, int dy, int dz, int nx, int ny, int nz,  
    int time, DicomPixelBuffer pixels, DicomPixelInterpolator interp);

The inputs \{x, y, z\} refer to voxel indices, and time refers to the time instance index, starting at zero. The four voxel dimensions of the image can be queried with: getNumCols() getNumRows(), getNumSlices(), and getNumTimes().

The DicomImage also contains spatial transform information for converting between voxel indices and patient-centered spatial locations. The affine transform can be acquired with the method getPixelTransform(). This allows the image to be placed in the appropriate 3D location, to correspond with any derived data such as segmentations. The spatial transformation is automatically extracted from the DICOM header information embedded in the files.

### 10.3 Loading a DicomImage

DICOM files and folders are read using the DicomReader class. The reader populates a supplied DicomImage with slices, forming the full 3D(+time) image. The basic pattern is as follows:

```java
String DICOM_directory = ... // define directory of interest
DicomReader reader = new DicomReader(); // create a new reader

// read all files in a directory, returning a newly constructed image
DicomImage image = reader.read(null, DICOM_directory);
```

The first argument in the read(...) command is an existing image in which to append slices. In this case, we pass in null to signal that a new image is to be created.

In some cases, we might wish to exclude certain files, such as meta-data files that happen to be in the DICOM folder. By default, the reader attempts to read all files in a given directory, and will print out an error message for those it fails to detect as being in a valid DICOM format. To limit the files to be considered, we allow the specification of a Java Pattern, which will test each filename against a regular expression. Only files with names that match the pattern will be included. For example, in the following, we limit the reader to files ending with the “dcm” extension.

```java
String DICOM_directory = ... // define directory of interest
DicomReader reader = new DicomReader(); // create a new reader
Pattern dcmPattern = Pattern.compile(".*\.dcm"); // files ending with .dcm

// read all files in a directory, returning a newly constructed image
DicomImage image = reader.read(null, DICOM_directory, dcmPattern, /* subdirs*/ false) ;
```

The pattern is applied to the absolute filename, with either windows and mac/linux file separators (both are checked against the regular expression). The method also has an option to recursively search for files in subdirectories. If the full list of files is known, then one can use the method:

```java
public DicomImage read(DicomImage im, List<File> files);
```

which will load all specified files.
10.3.1 Time-dependent images

In most cases, time-dependent images will be properly assembled using the previously mentioned methods in the DicomReader. Each slice should have a temporal position identifier that allows for the separate image stacks to be separated. However, we have found in practice that at times, the temporal position identifier is omitted. Instead, each stack might be stored in a separate DICOM folder. For this reason, additional read methods have been added that allow manual specification of the time index:

```java
public DicomImage read(DicomImage im, List<File> files, int temporalPosition);
public DicomImage read(DicomImage im, String directory, Pattern filePattern, boolean checkSubdirectories, int temporalPosition);
```

If the supplied `temporalPosition` is non-negative, then the temporal position of all included files will be manually set to that value. If negative, then the method will attempt to read the temporal position from the DICOM header information. If no such information is available, then the reader will guess the temporal position to be one past the last temporal position in the original image stack (or 0 if `im == null`). For example, if the original image has temporal positions `{0, 1, 2}`, then all appended slices will have a temporal position of three.

10.3.2 Image formats

The DicomReader attempts to automatically decode any pixel information embedded in the DICOM files. Unfortunately, there are virtually an unlimited number of image formats allowed in DICOM, so there is no way to include native support to decode all of them. By default, the reader can handle raw pixels, and any image format supported by Java’s ImageIO framework, which includes JPEG, PNG, BMP, WBMP, and GIF. Many medical images, however, rely on lossless or near-lossless encoding, such as lossless JPEG, JPEG 2000, or TIFF. For these formats, we provide an interface that interacts with the third-party command-line utilities provided by ImageMagick (http://www.imagemagick.org). To enable this interface, the ImageMagick utilities `identify` and `convert` must be available and exist somewhere on the system’s PATH environment variable.

ImageMagick Installation

To enable ImageMagick decoding, required for image formats not natively supported by Java (e.g. JPEG 2000, TIFF), download and install the ImageMagick command-line utilities from:

http://www.imagemagick.org/script/binary-releases.php

The install path must also be added to your system’s PATH environment variable so that ArtiSynth can locate the `identify` and `convert` utilities.

10.4 The DicomViewer

Once a DicomImage is loaded, it can be displayed in a model by using the DicomViewer component. The viewer has several key properties:

- **name**
  - the name of the viewer component

- **x, y, z**
  - the normalized slice positions, in the range [0,1], at which to display image planes

- **timeIndex**
  - the temporal position (image stack) to display

- **transform**
  - an affine transformation to apply to the image (on top of the voxel-to-spatial transform extracted from the DICOM file)

- **drawYZ**
  - draw the YZ plane, corresponding to position x
drawXZ
draw the XZ plane, corresponding to position y

drawXY
draw the XY plane, corresponding to position z

drawBox
draw the 3D image’s bounding box

pixelConverter
the interpolator responsible for converting pixels decoded in the DICOM slices into values appropriate for display. The converter has additional properties:

window
name of a preset window for linear interpolation of intensities

center
center intensity

width
width of window

Each property has a corresponding `getXxx(...)` and `setXxx(...)` method that can adjust the settings in code. They can also be modified directly in the ArtiSynth GUI. The last property, the `pixelConverter` allows for shifting and scaling intensity values for display. By default a set of intensity ‘windows’ are loaded directly from the DICOM file. Each window has a name, and defines a center and width used for linearly scale the intensity range. In addition to the windows extracted from the DICOM, two new windows are added: `FULL_DYNAMIC`, corresponding to the entire intensity range of the image; and `CUSTOM`, which allows for custom specification of the window center and width properties.

To add a `DicomViewer` to the model, create the viewer by supplying a component name and reference to a `DicomImage`, then add it as a `Renderable` to the `RootModel`:

```java
DicomViewer viewer = new DicomViewer("my image", dicomImage);
addRenderable(viewer);
```

The image will automatically be displayed in the patient-centered coordinates loaded from the `DicomImage`. In addition to this basic construction, there are convenience constructors to avoid the need for a `DicomReader` for simple DICOM files:

```java
// loads all matching DICOM files to create a new image
public DicomViewer(String name, String imagePath, Pattern filePattern, boolean <- checkSubdirs);
// loads a list of DICOM files to create a new image
public DicomViewer(String name, List<File> files);
```

These constructors generate a new `DicomImage` internal to the viewer. The image can be retrieved from the viewer using the `getImage()` method.

## 10.5 DICOM example

Some examples of DICOM use can be found in the `artisynth.core.demos.dicom` package. The model `DicomTest` loads a partial image of a heart, which is initially downloaded from the ArtiSynth website:

```java
package artisynth.demos.dicom;

import java.awt.Color;
import java.io.File;
import java.io.IOException;
import artisynth.core.renderables.DicomViewer;
import artisynth.core.workspace.DriverInterface;
```
import artisynth.core.workspace.RootModel;
import maspack.fileutil.FileManager;
import maspack.util.PathFinder;

public class DicomTest extends RootModel {

    // Dicom file name and URL from which to load it
    String dicom_file = "MR-MONO2-8-16x-heart";
    String dicom_url = "https://www.artisynth.org/files/data/dicom/MR-MONO2-8-16x-heart.gz";

    public void build(String[] args) throws IOException {
        // cache image in a local directory 'data' beneath Java source
        String localDir = PathFinder.getSourceRelativePath(this, "data/MONO2_HEART");
        // create a file manager to get the file and download it if necessary
        FileManager fileManager = new FileManager(localDir, "gz:" + dicom_url + "!/");
        fileManager.setConsoleProgressPrinting(true);
        fileManager.setOptions(FileManager.DOWNLOAD_ZIP); // download zip file first
        // get the file from local directory, downloading first if needed
        File dicomPath = fileManager.get(dicom_file);
        // create a DicomViewer for the file
        DicomViewer dcp = new DicomViewer("Heart", dicomPath.getAbsolutePath(), null, /* check subdirectories*/false);
        addRenderable(dcp); // add it to root model's list of renderable
    }
}

Lines 23-28 are responsible for downloading and extracting the sample DICOM zip file. In the end, dicomPath contains a reference to the desired DICOM file on the local system, which is used to create a viewer on line 34. We then add the viewer to the model for display purposes.

To run this example in ArtiSynth, select All demos > dicom > DicomTest from the Models menu. The model should load and initially appear as in Figure 10.3.

Figure 10.3: DICOM viewer image from DicomTest
Appendix A

Mathematical Review

This appendix reviews some of the mathematical concepts used in this manual.

A.1 Rotation transforms

Rotation matrices are used to describe the orientation of 3D coordinate frames in space, and to transform vectors between these coordinate frames.

Consider two 3D coordinate frames A and B that are rotated with respect to each other (Figure A.1). The orientation of B with respect to A can be described by a $3 \times 3$ rotation matrix $R_{BA}$, whose columns are the unit vectors giving the directions of the rotated axes $x'$, $y'$, and $z'$ of B with respect to A.

$R_{BA}$ is an orthogonal matrix, meaning that its columns are both perpendicular and mutually orthogonal, so that

$$R_{BA}^T R_{BA} = I \quad \text{(A.1)}$$

where $I$ is the $3 \times 3$ identity matrix. The inverse of $R_{BA}$ is hence equal to its transpose:

$$R_{BA}^{-1} = R_{BA}^T \quad \text{(A.2)}$$

Because $R_{BA}$ is orthogonal, $| \det R_{BA} | = 1$, and because it is a rotation, $\det R_{BA} = 1$ (the other case, where $\det R_{BA} = -1$, is not a rotation but a reflection). The 6 orthogonality constraints associated with a rotation matrix mean that in spite of having 9 numbers, the matrix only has 3 degrees of freedom.

Now, assume we have a 3D vector $v$, and consider its coordinates with respect to both frames A and B. Where necessary, we use a preceding superscript to indicate the coordinate frame with respect to which a quantity is described, so that $A^v$ and $B^v$ denote $v$ with respect to frames A and B, respectively. Given the definition of $R_{AB}$ given above, it is fairly straightforward to show that

$$A^v = R_{BA} B^v \quad \text{(A.3)}$$

and, given (A.2), that

$$B^v = R_{BA}^T A^v \quad \text{(A.4)}$$

Hence in addition to describing the orientation of B with respect to A, $R_{BA}$ is also a transformation matrix that maps vectors in B to vectors in A.

It is straightforward to show that

$$R_{BA}^{-1} = R_{BA}^T = R_{AB} \quad \text{(A.5)}$$

A simple rotation by an angle $\theta$ about one of the basic coordinate axes is known as a basic rotation. The three basic rotations about x, y, and z are:

$$R_x(\theta) = \begin{pmatrix} 1 & 0 & 0 \\ 0 & \cos(\theta) & -\sin(\theta) \\ 0 & \sin(\theta) & \cos(\theta) \end{pmatrix},$$

$$R_y(\theta) = \begin{pmatrix} \cos(\theta) & 0 & \sin(\theta) \\ 0 & 1 & 0 \\ -\sin(\theta) & 0 & \cos(\theta) \end{pmatrix},$$

$$R_z(\theta) = \begin{pmatrix} \cos(\theta) & -\sin(\theta) & 0 \\ \sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{pmatrix}.$$
Figure A.1: Two coordinate frames A and B rotated with respect to each other.

Figure A.2: Schematic illustration of three coordinate frames A, B, and C and the rotational transforms relating them.

\[
R_z(\theta) = \begin{pmatrix}
\cos(\theta) & -\sin(\theta) & 0 \\
\sin(\theta) & \cos(\theta) & 0 \\
0 & 0 & 1
\end{pmatrix}.
\]

Next, we consider transform composition. Suppose we have three coordinate frames, A, B, and C, whose orientation are related to each other by \( R_{BA}, R_{CB}, \) and \( R_{CA} \) (Figure A.6). If we know \( R_{BA} \) and \( R_{CA} \), then we can determine \( R_{CB} \) from

\[
R_{CB} = R_{BA}^{-1} R_{CA}.
\]

This can be understood in terms of vector transforms. \( R_{CB} \) transforms a vector from C to B, which is equivalent to first transforming from C to A,

\[
A_v = R_{CA} C_v,
\]

and then transforming from A to B:

\[
B_v = R_{BA}^{-1} A_v = R_{BA}^{-1} R_{CA} C_v = R_{CB} C_v.
\]

Note also from (A.5) that \( R_{CB} \) can be expressed as

\[
R_{CB} = R_{AB} R_{CA}.
\]

In addition to specifying rotation matrix components explicitly, there are numerous other ways to describe a rotation. Three of the most common are:

**Roll-pitch-yaw angles**
There are 6 variations of roll-pitch-yaw angles. The one used in ArtiSynth corresponds to older robotics texts (e.g., Paul, Spong) and consists of a roll rotation $r$ about the z axis, followed by a pitch rotation $p$ about the new y axis, followed by a yaw rotation $y$ about the new x axis. The net rotation can be expressed by the following product of basic rotations: $R_z(r)R_y(p)R_x(y)$.

**Axis-angle**

An axis angle rotation parameterizes a rotation as a rotation by an angle $\theta$ about a specific axis $u$. Any rotation can be represented in such a way as a consequence of Euler’s rotation theorem.

**Euler angles**

There are 6 variations of Euler angles. The one used in ArtiSynth consists of a rotation $\phi$ about the z axis, followed by a rotation $\theta$ about the new y axis, followed by a rotation $\psi$ about the new z axis. The net rotation can be expressed by the following product of basic rotations: $R_z(\phi)R_y(\theta)R_z(\psi)$.

### A.2 Rigid transforms

Rigid transforms are used to specify both the transformation of points and vectors between coordinate frames, as well as the relative position and orientation between coordinate frames.

Consider two 3D coordinate frames in space, A and B (Figure A.3). The translational position of B with respect to A can be described by a vector $p_{BA}$ from the origin of A to the origin of B (described with respect to frame A). Meanwhile, the orientation of B with respect to A can be described by the $3 \times 3$ rotation matrix $R_{BA}$ (Section A.1). The combined position and orientation of B with respect to A is known as the pose of B with respect to A.

Now, assume we have a 3D point $q$, and consider its coordinates with respect to both frames A and B (Figure A.4). Given the pose descriptions given above, it is fairly straightforward to show that

$$^Aq = R_{BA}^Bq + p_{BA}, \quad \text{(A.10)}$$

and, given (A.2), that

$$^Bq = R_{BA}^T(^AQ - p_{BA}). \quad \text{(A.11)}$$

If we extend our points into a 4D homogeneous coordinate space with the fourth coordinate $w$ equal to 1, i.e.,

$$q^* \equiv \begin{pmatrix} q \\ 1 \end{pmatrix}, \quad \text{(A.12)}$$

then (A.10) and (A.11) can be simplified to

$$^Aq^* = T_{BA}^Bq^* \quad \text{and} \quad ^Bq^* = T_{BA}^{-1}^Aq^*$$
Figure A.4: Point vectors $^A q$ and $^B q$ describing the position of a point $q$ with respect to frames A and B.

Figure A.5: The transform matrix $T_{BA}$ from B to A.

where

$$T_{BA} = \begin{pmatrix} R_{BA} & p_{BA} \\ 0 & 1 \end{pmatrix} \quad (A.13)$$

and

$$T_{BA}^{-1} = \begin{pmatrix} R_{BA}^T & -R_{BA}^T p_{BA} \\ 0 & 1 \end{pmatrix}. \quad (A.14)$$

$T_{BA}$ is the $4 \times 4$ rigid transform matrix that transforms points from B to A and also describes the pose of B with respect to A (Figure A.5).

It is straightforward to show that $R_{BA}^T$ and $-R_{BA}^T p_{BA}$ describe the orientation and position of A with respect to B, and so therefore

$$T_{BA}^{-1} = T_{AB}. \quad (A.15)$$

Note that if we are transforming a vector $v$ instead of a point between B and A, then we are only concerned about relative orientation and the vector transforms (A.3) and (A.4) should be used instead. However, we can express these using $T_{BA}$ if we embed vectors in a homogeneous coordinate space with the fourth coordinate $w$ equal to 0, i.e.,

$$v^* \equiv \begin{pmatrix} v \\ 0 \end{pmatrix}, \quad (A.16)$$

so that

$$^B v^* = T_{BA} ^A v^* \quad \text{and} \quad ^A v^* = T_{BA}^{-1} ^B v^*.$$ 

Finally, we consider transform composition. Suppose we have three coordinate frames, A, B, and C, each related to the other by transforms $T_{BA}$, $T_{CB}$, and $T_{CA}$ (Figure A.6). Using the same reasoning used to derive (A.6) and (A.9), it is easy to show that

$$T_{CB} = T_{BA}^{-1} T_{CA} = T_{AB} T_{CA}. \quad (A.17)$$
A.3 Affine transforms

An affine transform is a generalization of a rigid transform, in which the rotational component \( R \) is replaced by a general \( 3 \times 3 \) matrix \( A \). This means that an affine transform implements a generalized basis transformation combined with an offset of the origin (Figure A.7). As with \( R \) for rigid transforms, the columns of \( A \) still describe the transformed basis vectors \( x', y', \) and \( z' \), but these are generally no longer orthonormal.

Expressed in terms of homogeneous coordinates, the affine transform \( X_{BA} \) takes the form

\[
X_{BA} = \begin{pmatrix}
A_{BA} & p_{BA} \\
0 & 1
\end{pmatrix}
\]  

(A.18)

with

\[
X_{BA}^{-1} = \begin{pmatrix}
A_{BA}^{-1} & -A_{BA}^{-1}p_{BA} \\
0 & 1
\end{pmatrix}.
\]

(A.19)

As with rigid transforms, when an affine transform is applied to a vector instead of a point, only the matrix \( A \) is applied and the translation component \( p \) is ignored.

Affine transforms are typically used to effect transformations that require stretching and shearing of a coordinate frame. By the polar decomposition theorem, \( A \) can be factored into a regular rotation \( R \) plus a symmetric shearing/scaling matrix \( P \):

\[
A = RP
\]

(A.20)
Affine transforms can also be used to perform reflections, in which \( A \) is orthogonal (so that \( A^T A = I \)) but with \( \det A = -1 \).

### A.4 Rotational velocity

![Frame B rotating with respect to frame A.](image)

Given two 3D coordinate frames \( A \) and \( B \), the rotational, or angular, velocity of \( B \) with respect to \( A \) is given by a 3D vector \( \omega_{BA} \) (Figure A.8). \( \omega_{BA} \) is related to the derivative of \( R_{BA} \) by

\[
R_{BA} = \left[^A \omega_{BA}\right] R_{BA} = R_{BA} \left[^B \omega_{BA}\right]
\]  

(A.21)

where \( ^A \omega_{BA} \) and \( ^B \omega_{BA} \) indicate \( \omega_{BA} \) with respect to frames \( A \) and \( B \) and \( [\omega] \) denotes the 3 \( \times \) 3 cross product matrix

\[
[\omega] = \begin{pmatrix}
0 & -\omega_z & \omega_y \\
\omega_z & 0 & -\omega_x \\
-\omega_y & \omega_x & 0
\end{pmatrix}.
\]  

(A.22)

If we consider instead the velocity of \( A \) with respect to \( B \), it is straightforward to show that

\[
\omega_{AB} = -\omega_{BA}.
\]  

(A.23)

### A.5 Spatial velocities and forces

Given two 3D coordinate frames \( A \) and \( B \), the spatial velocity, or twist, \( \hat{v}_{BA} \) of \( B \) with respect to \( A \) is given by the 6D composition of the translational velocity \( v_{BA} \) of the origin of \( B \) with respect to \( A \) and the angular velocity \( \omega_{BA} \):

\[
\hat{v}_{BA} \equiv \begin{pmatrix} v_{BA} \\ \omega_{BA} \end{pmatrix}.
\]  

(A.24)

Similarly, the spatial force, or wrench, \( \hat{f}_B \) acting on a frame \( B \) is given by the 6D composition of the translational force \( f_B \) acting on the frame’s origin and the moment \( \tau \), or torque, acting through the frame’s origin:

\[
\hat{f}_B \equiv \begin{pmatrix} f_B \\ \tau_B \end{pmatrix}.
\]  

(A.25)

If we have two frames \( A \) and \( B \) rigidly connected within a rigid body (Figure A.9), and we know the spatial velocity \( \hat{v}_{BC} \) of \( B \) with respect to some third frame \( C \), we may wish to know the spatial velocity \( \hat{v}_{AC} \) of \( A \) with respect to \( C \). The
Figure A.9: Two frames A and B rigidly connected within a rigid body and moving with respect to a third frame C.

Angular velocity components are the same, but the translational velocity components are coupled by the angular velocity and the offset $p_{BA}$ between A and B, so that

$$v_{AC} = v_{BC} + p_{BA} \times \omega_{BC}. $$

$\hat{v}_{AC}$ is hence related to $\hat{v}_{BC}$ via

$$
\begin{pmatrix}
  \dot{v}_{AC} \\
  \dot{\omega}_{AC}
\end{pmatrix} =
\begin{pmatrix}
  I & [p_{BA}] \\
  0 & I
\end{pmatrix}
\begin{pmatrix}
  \dot{v}_{BC} \\
  \dot{\omega}_{BC}
\end{pmatrix},
$$

where $[p_{BA}]$ is defined by (A.22).

The above equation assumes that all quantities are expressed with respect to the same coordinate frame. If we instead consider $\hat{v}_{AC}$ and $\hat{v}_{BC}$ to be represented in frames A and B, respectively, then we can show that

$$A^A \hat{v}_{AC} = X_{BA}^B \hat{v}_{BC}.$$  \hspace{1cm} (A.26)

where

$$X_{BA} \equiv \begin{pmatrix}
  R_{BA} & [p_{BA}] R_{BA} \\
  0 & R_{BA}
\end{pmatrix}. $$  \hspace{1cm} (A.27)

The transform $X_{BA}$ is easily formed from the components of the rigid transform $T_{BA}$ relating B to A.

The spatial forces $\hat{f}_A$ and $\hat{f}_B$ acting on frames A and B within a rigid body are related in a similar way, only with spatial forces, it is the moment that is coupled through the moment arm created by $p_{BA}$, so that

$$\tau_A = \tau_B + p_{BA} \times \hat{f}_B. $$

If we again assume that $\hat{f}_A$ and $\hat{f}_B$ are expressed in frames A and B, we can show that

$$A^A \hat{f}_A = X_{BA}^B \hat{f}_B,$$  \hspace{1cm} (A.28)

where

$$X_{BA} \equiv \begin{pmatrix}
  R_{BA} & 0 \\
[0_{BA}] R_{BA} & R_{BA}
\end{pmatrix}. $$  \hspace{1cm} (A.29)

### A.6 Spatial inertia

Assume we have a rigid body with mass $m$ and a coordinate frame located at the body’s center of mass. If $v$ and $\omega$ give the translational and rotational velocity of the coordinate frame, then the body’s linear and angular momentum $p$ and $L$ are given by

$$p = m v \quad \text{and} \quad L = J \omega,$$  \hspace{1cm} (A.30)
where \( \mathbf{J} \) is the \(3 \times 3\) rotational inertia with respect to the center of mass. These relationships can be combined into a single equation

\[
\dot{\mathbf{p}} = \mathbf{M} \dot{\mathbf{v}},
\]  

(A.31)

where \( \dot{\mathbf{p}} \) is the spatial momentum and \( \mathbf{M} \) is a \(6 \times 6\) matrix representing the spatial inertia:

\[
\dot{\mathbf{p}} \equiv \begin{pmatrix} \mathbf{p} \\ \mathbf{L} \end{pmatrix}, \quad \mathbf{M} \equiv \begin{pmatrix} \mathbf{mI} & 0 \\ 0 & \mathbf{J} \end{pmatrix}.
\]  

(A.32)

The spatial momentum satisfies Newton’s second law, so that

\[
\dot{\mathbf{f}} = \frac{d\dot{\mathbf{p}}}{dt} = \mathbf{M} \frac{d\dot{\mathbf{v}}}{dt} + \dot{\mathbf{M}} \dot{\mathbf{v}},
\]  

(A.33)

which can be used to find the acceleration of a body in response to a spatial force.

When the body coordinate frame is not located at the center of mass, then the spatial inertia assumes the more complicated form

\[
\begin{pmatrix} \mathbf{mI} & -m[c] \\ m[c] & \mathbf{J} - m[c][c] \end{pmatrix},
\]  

(A.34)

where \( c \) is the center of mass and \([c]\) is defined by (A.22).

Like the rotational inertia, the spatial inertia is always symmetric positive definite if \( m > 0 \).
References


