



FEBio Studio

Finite Elements for Biomechanics

User's Manual Version 1.0

Last Updated: May 26, 2020

Contributors:

- Dr. Steve Maas (steve.maas@utah.edu)
- Dr. Gerard Ateshian (ateshian@columbia.edu)
- Dr. Jeff Weiss (jeff.weiss@utah.edu)
- Dave Rawlins (rawlins@sci.utah.edu)

Contact Address:

Musculoskeletal Research Laboratories, University of Utah
72 S. Central Campus Drive, Room 2646
Salt Lake City, Utah

Website

MRL: <http://mrl.sci.utah.edu>
FEBio: <http://febio.org>

Forum

<https://forums.febio.org/>

Acknowledgments

Development of the FEBio project is supported in part by a grant from the U.S. National Institutes of Health (R01GM083925).



Contents

1	Introduction	6
1.1	Overview of PreView	6
1.2	About this document	6
2	Getting Started	8
2.1	Tutorial 1: Navigating the PreView GUI	8
2.1.1	Step 1: An overview of PreView	8
2.1.2	Step 2: Opening a file	8
2.1.3	Step 3: Navigating the Graphics View	10
2.1.4	Step 4: Selecting objects	10
2.1.5	Step 5: Transforming an object	10
2.1.6	Step 6: Exporting the Model	11
2.2	Tutorial 2: A simple tensile test	12
2.2.1	Step 1: Creating the geometry	12
2.2.2	Step 2: Setting up the boundary conditions	12
2.2.3	Step 3: Creating and assigning materials	13
2.2.4	Step 4: Creating the Mesh	14
2.2.5	Step 5: Setting up the analysis	14
2.3	Tutorial 3: Twisted bar problem	16
2.3.1	Step 1: Creating the geometry	16
2.3.2	Step 2: Setting up the boundary conditions	17
2.3.3	Step 3: Setting up the materials	17
2.3.4	Step 4: Creating the mesh	18
2.3.5	Step 5: Defining the analysis step	18
2.4	Tutorial 4: The billet problem	19
2.4.1	Step 1: Creating the geometry	19
2.4.2	Step 2: Setting up the boundary conditions	20
2.4.3	Step 3: Setting up the materials	22
2.4.4	Step 4: Setting up the analysis	22
2.5	Tutorial 5: Biphasic unconfined compression	23
2.5.1	Step 1: Creating the geometry	23
2.5.2	Step 2: Setting up the boundary conditions	24
2.5.3	Step 3: Defining material parameters	25
2.5.4	Step 4: Setting up the analysis	25
2.6	Tutorial 6: A multi-step analysis	26
2.6.1	Step 1: Creating the geometry	26
2.6.2	Step 2: Defining the analysis steps	26

2.6.3 Step 3: Setting up the boundary conditions	27
2.6.4 Step 4: Defining material parameters	28
2.6.5 Step 5: Creating the mesh	28
2.7 Tutorial 7: Heat Transfer Analysis	29
2.7.1 Step 1: Geometry and Meshing	29
2.7.2 Step 2: Choosing a material	29
2.7.3 Step 3: Boundary Conditions	30
2.7.4 Step 4: Defining a heat transfer analysis	30
2.8 Tutorial 8: Reversible binding kinetics	31
2.8.1 Step 1: Defining the model data	31
2.8.2 Step 1: Defining the solute and solid-bound molecules	31
2.8.3 Step 2: Creating a multiphasic material	32
2.8.4 Step 3: Creating a chemical reaction	32
2.8.5 Step 4: Geometry and meshing	33
2.8.6 Step 5: Boundary and initial conditions	33
2.8.7 Step 6: Setting up the analysis	34
2.9 Tutorial 9: Axisymmetric analysis of biphasic indentation	36
2.9.1 Creating the geometry	36
2.9.2 Setting up the materials	38
2.9.3 Setting up the boundary conditions	38
2.9.4 Defining the analysis step	39
2.10 Tutorial 10: Setting up a model from imported geometry	40
2.10.1 Step 1: Importing the mesh	40
2.10.2 Step 2: Setting up boundary conditions, method 1	40
2.10.3 Step 2: Setting up boundary conditions, method 2	41
2.10.4 Step 3: Finishing the model	42
2.11 Tutorial 11: Setting up a Fluid analysis	43
2.11.1 Step 1: Creating the Geometry and Mesh	43
2.11.2 Step 2: Setting up the materials	44
2.11.3 Step 3: Setting up the boundary conditions	44
2.11.4 Step 4: Setting up the analysis step	45
2.12 Tutorial 12: Setting up a Fluid-Structure Interaction Analysis	46
2.12.1 Step 1: Creating the Geometry and Mesh	46
2.12.2 Step 2: Setting up the materials	47
2.12.3 Step 3: Setting up the boundary conditions	48
2.12.4 Step 4: Setting up the analysis step	50
3 The PreView Environment	52
3.1 The Graphical User Interface	52
3.1.1 Overview	52
3.1.2 Navigating the GUI	52
3.2 The Menu Bar	53
3.2.1 The File Menu	53
3.2.2 The Edit Menu	54
3.2.3 The Physics Menu	55
3.2.4 The Tools Menu	55
3.2.5 The View Menu	56
3.2.6 The Help Menu	56

3.3	The Main Tool Bar	57
3.4	The Graphics View	58
3.5	The Graphics Toolbar	59
3.6	The Model Viewer	59
3.6.1	The Search Panel	62
3.7	The Model Viewer Panels	62
3.7.1	Editing Selections	62
3.8	The Build Panel	63
3.8.1	The Create Panel	63
3.8.2	The Edit Panel	63
3.8.3	The Mesh Panel	63
3.8.4	The Tools Panel	63
3.9	The Curve Editor	63
3.10	The Mesh Inspector	66
3.11	PreView Options	66
4	Creating, Loading, and Saving Projects	68
4.1	Starting a new project	68
4.2	Loading a project	68
4.3	Saving a project	69
4.4	PreView Special File Formats	69
4.4.1	PreView Object File	69
4.4.2	PreView Material File	69
5	Creating and Editing Geometry	70
5.1	Creating Geometry	70
5.2	Importing Geometry	70
5.3	Editing Geometry	70
5.4	Creating and Editing a Mesh	71
5.4.1	Meshing Primitives	71
5.4.2	Editable Surfaces	71
5.4.3	Editable Meshes	72
6	Materials	74
6.1	Adding materials	74
6.2	Setting material parameters	74
6.3	Assigning materials	75
6.4	Creating a Solute Table	75
6.5	Creating a Solid-Bound Molecule Table	75
6.6	Adding Chemical Reactions	76
7	Boundary Conditions and Loads	77
7.1	Boundary Conditions	77
7.1.1	Fixed Displacement	78
7.1.2	Fixed Shell Displacement	78
7.1.3	Fixed Shell Rotation	78
7.1.4	Zero Fluid Pressure	78
7.1.5	Zero Temperature	78

7.1.6 Zero Solute Concentration	79
7.1.7 Prescribed Displacement	79
7.1.8 Prescribed Shell Rotation	79
7.1.9 Prescribed Shell Displacement	79
7.1.10 Prescribed Fluid Pressure	79
7.1.11 Prescribed Temperature	79
7.1.12 Prescribed Solute Concentration	79
7.2 Surface Loads	80
7.2.1 Pressure	80
7.2.2 Traction forces	80
7.3 Initial Conditions	81
7.3.1 Velocity	81
7.3.2 Shell Velocity	81
7.3.3 Temperature	81
7.3.4 Concentration	81
7.3.5 Initial Fluid Pressure	82
7.4 Assigning Boundary Conditions	82
8 Contact and Constraints	83
8.1 Rigid Body Constraints	83
8.2 Contact	83
8.2.1 Rigid Interfaces	85
8.2.2 Sliding Interfaces	85
8.2.3 Tied Interfaces	86
9 Defining Analysis Steps	87
9.1 The Initial Step	87
9.2 Adding an Analysis Step	87
9.2.1 Analysis	88
9.2.2 Time Settings	88
9.2.3 Nonlinear Solver	89
9.2.4 Linear Solver	89
10 Running FEBio from PreView	90
11 Mesh Import Formats	91
11.1 FEBio	91
11.2 NIKE3D	91
11.3 HyperMesh ASCII	92
11.4 ABAQUS	92
11.5 LSDYNA keyword	93
11.6 ANSYS	93
11.7 DXF	93
11.8 Hypersurface ASCII	94
11.9 GMsh	94
11.10 BYU format	94
11.11 VTK format	94

Chapter 1

Introduction

1.1 Overview of PreView

PreView is a finite element preprocessing software package. Although it has some mesh generation capabilities, its primary function is to set up the boundary and loading conditions, material properties, and analysis parameters for finite element analysis with the software FEBio. The features of PreView include:

- User-friendly UI that greatly facilitates the FE modeling process
- Primitive mesh generation, e.g. boxes, cylinders, spheres, etc.
- Tetrahedral mesh generation via TetGen
- Mesh editing on element and sub-element level
- Supports various FE file formats, including FEBio, Abaqus, Ansys, Comsol, LSDYNA
- Supports various mesh file formats, including STL, VTK, BYU.
- Supports most of FEBio's modeling capabilities, including boundary conditions, surface and body loads, contact interfaces, analysis types, and more.
- FE model can be exported to FEBio and NIKE3D file formats.
- Mesh can also be exported to various file formats, including LSDYNA, VTK, and more.

PreView has been designed to create FEBio input files. Although it can create input files for other FE programs (like NIKE3D), this document assumes that FEBio will be used as the finite element solver. It is important to keep this in mind, since other programs may not support all the features of FEBio.

1.2 About this document

This document is the User's Manual to PreView. Although PreView has been designed for FEBio, it does not describe the FEBio features in much detail. A more in-depth description of the FEBio features can be found in the *FEBio User's Manual* and the *FEBio Theory Manual*.

Chapter 2 introduces the capabilities of PreView through a set of tutorials. New users are advised to follow these tutorials to familiarize themselves with the components of PreView's graphical user interface and with the steps required to set up a well-defined FE problem.

Chapter 3 explains the different components of PreView's environment in detail. Users will find more detailed information on how to work with PreView's GUI that was introduced in the tutorials of Chapter 2.

Chapter 4 introduces users to the new project template features and discusses how new projects can be created, as well as how to save and open PreView project files.

Chapter 5 describes how to create geometry. This chapter illustrates how to build geometry with PreView's built-in mesh generation capabilities as well as how to import meshes from several other file formats.

Chapter 6 will show you how to add materials and assign them to the various parts of your model.

Chapter 7 describes how to setup the many different boundary and loading conditions, which are necessary for a well-defined finite element model.

Chapter 8 discusses some more advanced boundary conditions, how to connect rigid and deformable parts, as well as how to properly setup contact interfaces.

Chapter 9 describes how to setup an analysis in PreView, such as static or dynamic structural mechanics, biphasic, heat transfer and more.

Chapter 2

Getting Started

This chapter gives a brief tour of PreView through a set of tutorials. The reader will be introduced to the graphical user interface (GUI) and the different components of PreView. The reader will also learn a basic set of skills to build FE models.

2.1 Tutorial 1: Navigating the PreView GUI

In this tutorial we will familiarize the user with the most important components of the PreView GUI and how to navigate it.

2.1.1 Step 1: An overview of PreView

Figure 1 shows an overview of the components of PreView's GUI. The *Main menu* lists all the available menu items. The *Main Toolbar*, located directly below the menu, offers an alternative way to invoke some of these menu items. The *Model Viewer* shows an overview of the components of the model. The *Graphics View* covers the largest part of the GUI and shows a 3D view of the model. The *Graphics Toolbar* is located below the Graphics View and displays some information regarding the Graphics View. The *Build panel* is where most of PreView's primitive modeling and mesh editing tools can be found. It consists of several command panels, each of which offers several modeling and editing tools.

For instance, selecting the *Create* panel will show options for creating new geometry. The other panels are: *Edit* for changing object properties, *Mesh* for creating and editing finite element meshes, and *Tools* which lists some useful features. See Chapter 3 for a more detailed overview of the different components of the GUI.

Most of the panels in PreView are dockable panels that can be moved around and closed. If you close a panel, you can show it again by finding its name at the bottom of the *View* menu.

2.1.2 Step 2: Opening a file

Open the file *tutorial1.prv* from the *Examples* folder of your PreView installation folder. You can do this by selecting the *File\Open* menu item. A standard file open dialog box appears. Locate the file, select it and click on *Open*. The UI will now look something like Figure 2.2.

This file contains an already completed FE model that is ready to be exported. It is a simply beam bending analysis. However, before doing so, let's explore some of the PreView's GUI components.

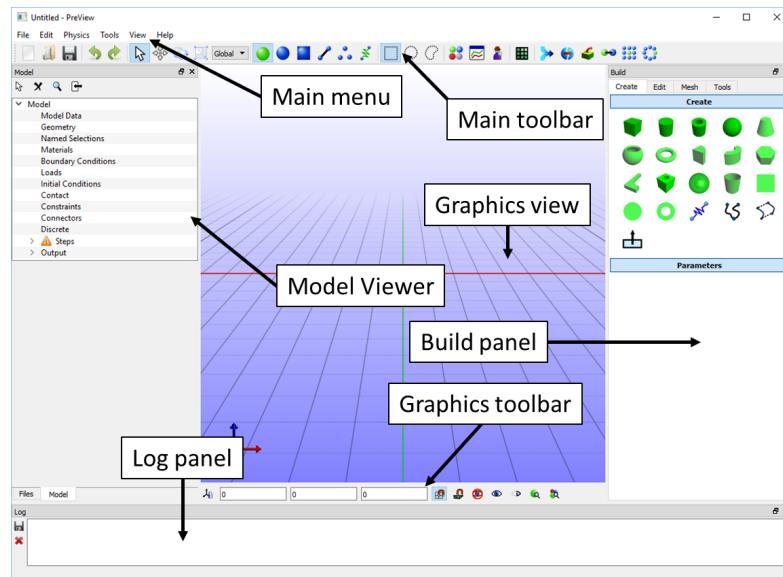


Figure 2.1: The main components of the PreView GU. The Main Menu provides access to most of PreView's functionality. The Main toolbar offers shortcuts for some commonly used features. The Graphics View shows a 3D rendering of the model. The Build panel shows all the geometry creation and editing tools. The Model Viewer displays a hierarchical overview of the finite element model.

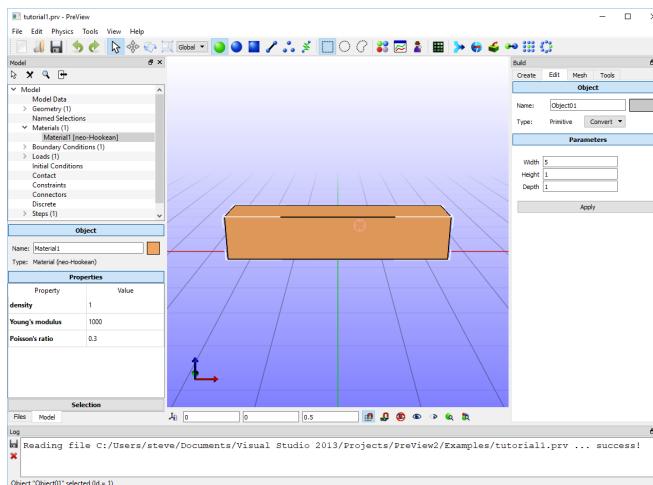


Figure 2.2: The UI after opening the tutorial1.prv file. The Model Viewer is now populated with the model components.

2.1.3 Step 3: Navigating the Graphics View

In this step we will show you the most important skills you need to navigate the Graphics View, namely rotating the view, zooming, panning, and selecting and transforming objects.

To rotate the view, click anywhere in an open area of the graphics view with the left mouse button and hold it down. By dragging the mouse left and right you can rotate the view to the left or to the right. Similarly, by dragging the mouse up and down you rotate the model up and down. Panning works similarly as rotating, except you use the middle mouse button instead of the left. Hold down the middle mouse button and drag the mouse to pan the view. You can zoom in or out by holding down the right mouse button. Drag the mouse up to zoom in and move it down to zoom out. See section 3.1.2 for an overview of the different methods of changing the view.

2.1.4 Step 4: Selecting objects

Selection is another important aspect for interacting with the model. Selection requires a selection context, which tells PreView what type of entity you would like to select. The current selection context is highlighted on the toolbar and can be changed by clicking on the corresponding selection button. For instance, to select an object, first click the “Select Object” button on the main toolbar (i.e. the button with the green ball). With the button selected, you can now select any object in the model by clicking on it with the left mouse button. Selecting another object will automatically unselect the first selected object. To add an object to the current selection, hold down the *shift* key when you select the object. Similarly, you can remove an object from the current selection by holding down the *ctrl* key when clicking on the object.

To select several objects with one motion, first choose a selection method from the main toolbar. There are three selection methods: rectangle, circle, and free form. For example, select the rectangle selection method. Now, hold down the *shift* key and then click and hold down the left mouse button on an empty part of the screen. With the *shift* key and left mouse button down, drag the mouse to draw a rectangle around the objects you wish to select. Once you let go of the left mouse button, all objects that are inside or intersect the rectangle will be selected. You can also use this procedure to unselect several objects at once. Just repeat the procedure but hold down the *ctrl* key instead of the *shift* key. The other selection methods work similarly.

2.1.5 Step 5: Transforming an object

Once an object is selected you can transform it, which means you can translate, rotate, or scale it. To apply a particular transformation, make sure one or more objects are selected. Then click on one of the transform buttons on the toolbar. For example, to translate an object click on the translate toolbar button (i.e. the button with the crossed arrows). A triad appears on the selected object(s). The triad is positioned at the current pivot point, which is the center from which the transformation is applied. The triad consists of three colored arms and three colored planes. When you move the mouse over one of the arms or planes, it becomes highlighted. A highlighted arm or plane can be selected and when you drag the mouse, you can move the object in the corresponding direction.

For more precise modeling you can also enter the transformation numerically using the *Transform* dialog which is accessible from the *Edit* menu (*Edit\Transform...*).

If you accidentally moved an object, you can undo your action by selecting the *Edit\Undo* menu (shortcut *ctrl+z*). The *Edit\Redo* menu option allows you to redo your last undone action. Note that you have a virtually unlimited undo-redo stack so you rarely have to worry about making

mistakes. However, for large models the undo stack may consume a lot of memory. The undo stack is cleared each time you save your model.

2.1.6 Step 6: Exporting the Model

The model that was loaded for this tutorial is ready to be used. We can therefore export it to a file format that can be read by your favorite finite element solver (e.g. FEBio). To export the model, select the *File\Export FE model* menu. A standard Save dialog box appears. Select a file format from the filter menu, enter a name for the file, and click on *Save*.

2.2 Tutorial 2: A simple tensile test

In this tutorial a simple FE model of a tensile test will be constructed. A rectangular box will be created and then the appropriate fixed and prescribed displacements will be applied to pull on one end of the box. This tutorial will also give an outline of the different steps that are required to build a well-defined FE problem.

2.2.1 Step 1: Creating the geometry

The geometry for this first problem is quite simple, consisting of a single rectangular box. To create a box, select the *Create* tab from the *Build panel* (Figure 2.3). Click the *Box* button to create a simple rectangular box. If you're not quite sure which button it is, you can hover over a button with the mouse and a tool tip appears telling you what geometry that button will create. When you click on the *Box* button, the available creation parameters are displayed below.

The position coordinates of the box can be entered directly on the create panel. Alternatively, you can click anywhere on the grid to position the new box. Then, enter the following parameters, $width = 4$, $height = 1$, $depth = 1$, and click the *Create* button to create the box.

This completes the geometry creation step. The next section of the tutorial teaches you how to add the boundary conditions. In this tutorial we will apply some constraints and a prescribed displacement to the geometry.

2.2.2 Step 2: Setting up the boundary conditions

This tutorial requires only two simple boundary conditions. The left side of the box will be constrained from moving. A prescribed displacement will be applied to the right side of the box.

To apply a boundary condition, select the *Physics/Add Boundary Condition* menu. A dialog box appears that allows you to select the type of boundary condition. Select the *Fixed displacement*. Click *OK* to create the boundary condition. The boundary condition will now be visible in the *Model Viewer*. At the bottom of the *Model Viewer* several panes are now visible that allow you to customize this boundary condition. In the *Properties* pane check the x , y , and z degrees of freedom.

Next, we need to apply this constraint to the appropriate surface of the model. On the main toolbar, you will see that there are five selection icons. The first one (a green ball) is the object selection button. The next four allow you to select parts, faces, edges, and nodes, respectively. For our first boundary condition we will select the left face of our box. First click on the *Select Faces* icon on the toolbar (blue square). This will allow you to select any of the faces on the box. Click with the left mouse button on the left face of the box. It should become highlighted as shown in the figure below. Now notice the *Selection* pane on the bottom of the *Model Viewer*. This box will list all the surfaces (or edges or nodes) to which this boundary condition is applied. With the left face of the box highlighted, click on the plus sign in the selection box. This will add the current selection to the selection list of the boundary condition.

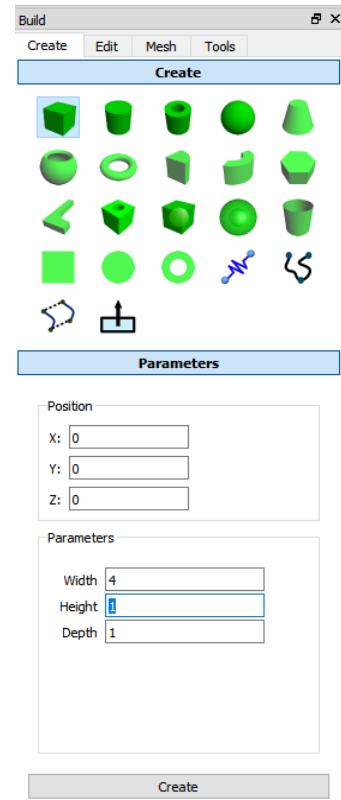


Figure 2.3: The Create panel is used to create various primitives.

Three more buttons are available in the selection box. The button with the minus sign subtracts the items that are currently selected in the Graphics View from the list. You can also select the lines in the selection box by clicking on them. You can select as many as you want. The button with the X-sign will remove all selected lines and the corresponding model components from the selection list. Finally, when you push the arrow button, the geometry items corresponding to the selected lines in the selection box will be selected in the Graphics View.

In a similar fashion, apply a fixed displacement to the right face of the box, but this time only constrain the y and z degrees of freedom.

Finally, we'll apply a prescribed displacement. Select the *Physics/Add Boundary Condition* menu. Now select *Prescribed displacement* from the list. In the properties box on the *Model Viewer*, make sure the *X-displacement* degree of freedom is selected and that the scale value is set to 1.0. Now assign the right face to this boundary condition using the same procedure that was used to assign faces to the fixed constraints. The right face will now have two boundary conditions applied to it: a fixed constraint for the y and z degrees of freedom and a prescribed constraint in the x degree of freedom. This concludes the boundary conditions.

2.2.3 Step 3: Creating and assigning materials

In this model, we will use a Mooney-Rivlin material which is an incompressible material model for rubber, but in biomechanics also often used to represent certain biological tissues. To add a material to the model, open the *Material Browser* by selecting the *Physics/Add Material* menu. Alternatively you can also open the *Material Browser* by right-clicking on the *Materials* item in the Model Viewer and selecting *Add Material* from the popup menu. There is also a shortcut for this on the toolbar (icon with four colored squares). First, from the Category option select "uncoupled elastic". Then, from the list of available material models, select the Mooney-Rivlin material and click *OK*. The newly added material should now be highlighted in the Model Viewer and the material properties are now displayed in the properties panel, located below the Model Viewer. Then, set the following material properties:

- density = 1.0
- c1 = 1.0
- c2 = 0.0

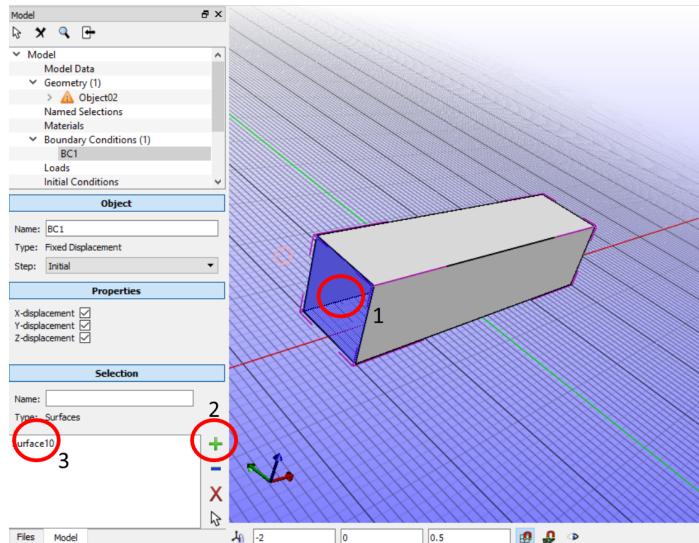


Figure 2.4: To apply a face to a boundary condition, 1) first, select the face by clicking on it in the graphics view, 2) click the + button in the selection box of the boundary condition, 3) The surface name will be shown in the list box. Multiple surfaces can be added. To remove a face, either select it in the graphics view and press the – button or select it in the list and press the X button.

- bulk modulus = 100

With the material selected in the Model Viewer you will notice, just like boundary conditions, it too has a selection box that lists the parts to which this material is assigned. After you have selected the object in the Graphics View, click on the plus-sign in the material's selection box, to assign this material to the object. The color of your object will now change to the same color of the material to indicate that it is now associated with that material.

2.2.4 Step 4: Creating the Mesh

We are almost ready with our first model. The final step is generating a mesh for our geometry. Select the box and then select *Mesh* from the build panel. The Mesh panel shows all parameters that will affect the meshing algorithm of the box. Set the following parameters and leave all other parameters set to their default values:

- Nx = 20
- Ny = 5
- Nz = 5

Next, click on the *Apply* button to generate the mesh. The view will update to show the new mesh. By default, PreView will not show the mesh lines. To show them select the *View/Toggle Mesh lines* menu or simply press the 'm' key.

2.2.5 Step 5: Setting up the analysis

Finally, we need to create an analysis step to define what type of analysis we want to do. Select *Physics/Add Step* from the menu. This will bring up a dialog box that displays the available analysis steps. Select the *Structural Mechanics* analysis and click Ok. The Step properties are now shown in the Model Viewer. The default settings are good enough for this first tutorial so no need to change any.

Note that in the Model Viewer, two steps appear in the list: the step you just created as well as a step called *Initial*. The initial step is not an analysis step (e.g. you cannot set time step settings for the initial step). Instead it is used to define the permanent boundary conditions, that is, the boundary conditions that

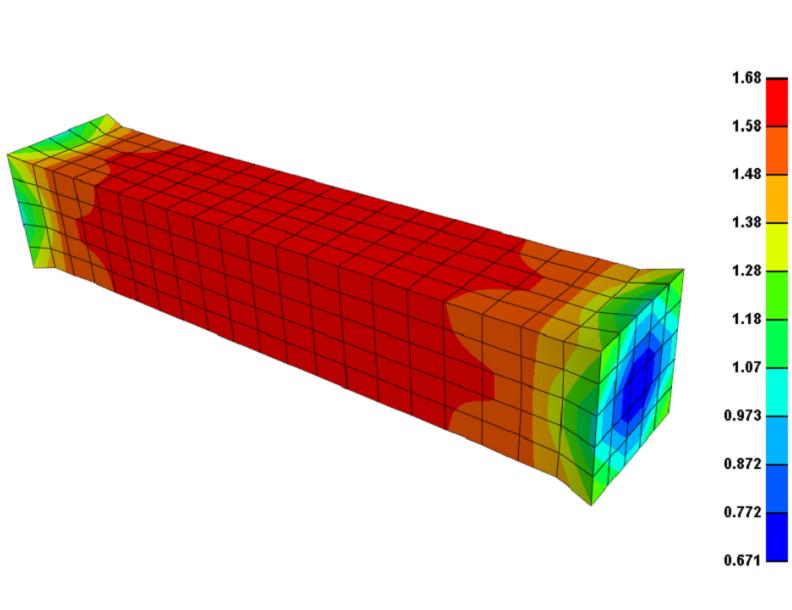


Figure 2.5: The completed model for tutorial 2. This figure shows the final deformed state. The color plot represents the Von Mises stress distribution of the final deformed state.

will propagate through all analysis steps. This aspect will be explained in more detail in the multi-step tutorial below. At this point it is a good idea to save your model. Select the *File/Save* menu to save your model.

This completes the tutorial. You now should be able to create some simple geometry, apply fixed and prescribed constraints, create materials and assign them to your geometry. You also learned how to modify the meshing parameters and define an analysis step. Figure 2.5 shows the solution for this problem when run in FEBio. It shows the deformed state as well as the von-mises stress.

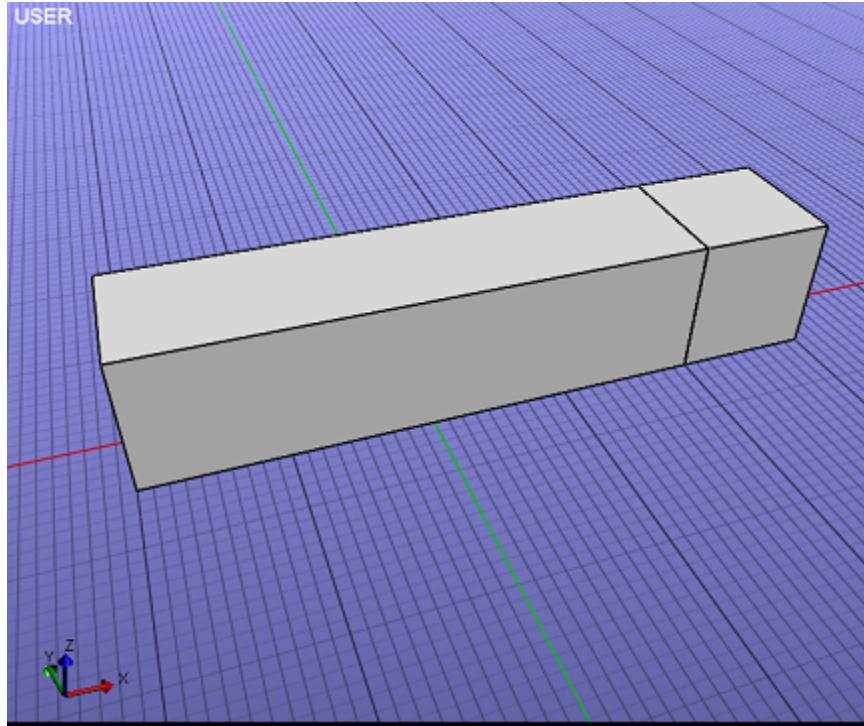


Figure 2.6: The model geometry for tutorial 3 consists of two boxes joined at one end.

2.3 Tutorial 3: Twisted bar problem

This tutorial will illustrate how to create a rigid body, apply prescribed rotations to a rigid body, and connect a deformable mesh to a rigid body.

2.3.1 Step 1: Creating the geometry

The geometry consists of two boxes: one for the deformable mesh and the other for the rigid body. We will create the deformable mesh first. Activate the *Create* panel and create a box located at the origin $(0,0,0)$ with parameters $\text{width} = 4$, $\text{height} = 1$, $\text{depth} = 1$. Next, create a second box located at $(2.5, 0, 0)$ and with parameters $\text{width} = 1$, $\text{height} = 1$, $\text{depth} = 1$. This will place the second box right next to the first one. Your model should now look something like Figure 2.6.

To attach the deformable mesh to the rigid body we need to select which face we wish to attach. To facilitate the selection we'll first hide the second, smaller box. You can hide an object by locating it in the *Model Viewer* and right-clicking on it. From the popup menu select *hide*. You'll notice that the name of the box is now grayed out in the *Model Viewer*. Now, with only the first box showing select the right face, that is, the face that is touching the second box (before you hid it). In order to refer to this selection later, we will name the selection using the *Edit/Name Selection* menu. You can enter a name for this selection or simply accept the default. The selection will now show up under the *Named Selection* item in the Model Viewer. Note that you can also name selections of parts, edges, and points. Named selections are convenient ways to refer to parts of your model easily. When you select a named selection in the Model Viewer, you will see that it has a selection box, which allows you to edit the named selection.

Before you continue, make sure you unhide the second box. To unhide it, locate it in the *Model*

Viewer, right-click on it and select the *Unhide* option from the popup menu.

2.3.2 Step 2: Setting up the boundary conditions

Select the face on the left end of the first box and apply a fixed constraint in the x , y , and z degrees of freedom. (See tutorial 1 for details on how to do this.) Next, we will attach the named selection that we created in the previous section to a rigid body. Before we can do that we need to add a rigid body to our model. Open the *Material Browser* using the *Physics/Add Material* menu, select the *rigid body* from the list (under the Other category) and click the *Add* button. The creation of a rigid material also implicitly creates a rigid body. An icon will show up in the Graphics View to indicate where the rigid body is located in space. Note that the rigid body does not have any geometry yet, but we'll deal with that later. The rigid body is initially placed at the origin. The location of the rigid body also defines its center of rotation. We would like to place this center of rotation in the center of the second box. To do this, select the rigid body material in the Model Viewer. In the properties list below the Model Viewer set the *center of mass* parameter to $2.5, 0, 0.5$. This places the rigid body in the correct spot.

Next, we'll create a rigid interface that connects a deformable body to a rigid body. Select the *Physics/Add Contact* menu and select *rigid* from the dialog box that appears. Click the *OK* button. A rigid interface only requires one parameter, namely the rigid body material. Select the rigid body you created earlier from the drop down list in the properties list. We still need to define to which part of the deformable box this rigid interface will be applied. Find the surface you defined earlier in the named selection item of the Model Viewer. Select it in the Model Viewer and click on the *Select* tool button at the top of the Model Viewer to select the surface in the Graphics View. With the surface now selected, select the rigid interface in the Model Viewer. In the selection box click on the '+' sign to assign the currently selected surface to the rigid interface. That's it! Your deformable mesh is now connected to the rigid body.

Note that it was not necessary to create a named selection. You could have selected the surface directly and assigned it to the rigid interface. However, it is often convenient to identify selections by name so they can be retrieved more easily later.

Our rigid body is initially unconstrained. To prevent it from flying off in space we'll have to constrain it. Go to the *Physics/Add Rigid Constraint* menu. In the dialog box that shows up select the rigid material and select the *fixed displacement/rotation* option. After you click *OK*, the constraints is selected in the Model Viewer. In the properties list check the all the boxes except the *x-rotation* box. We will apply a different constraint to the *x*-rotational degree of freedom.

To apply the second constraint, select the *Physics/Add Rigid Constraint* menu item again. This time select the *Prescribe Displacement/Rotation* option from the list. In the properties list select the *x-rotation* degree of freedom. Enter the value 3.14 in the value parameter.

2.3.3 Step 3: Setting up the materials

We already have created one material, namely the material for the rigid body. We now create the second material that we will assign to the deformable mesh. Open the *Material Browser* from the *Physics/Add Material* menu. Select a Mooney-Rivlin material and click the *Add* button to add it to the model. Set the following material parameters:

- density = 1
- c1 = 1

- $c_2 = 0$
- bulk modulus = 100

Note that the E and v parameters do not need to be set for the rigid body.

Now select the first, larger box in the Graphics View (make sure the second box is not selected and that you are at the *object* level) and assign the Mooney-Rivlin material to this box by clicking on the $+$ button in the material's selection box with the box selected in the graphics view. Now, select the second box and assign the rigid body material (Material 1) to this box.

2.3.4 Step 4: Creating the mesh

Next, we will create the mesh for the objects in the model. Select the deformable box (the one you created first and the larger of the two) and activate the *Mesh* panel. Enter the following settings:

- $N_x = 20$
- $N_y = 5$
- $N_z = 5$

Leave other parameters at their default value and click the *Apply* button. Repeat this for the second mesh and set the parameters as follows:

- $N_x = 1$
- $N_y = 1$
- $N_z = 1$

Note that this effectively creates a single element mesh for the rigid body. The completed model should now look something like the picture on the right.

It is also of some importance to point out that the rigid interface allows you to connect a rigid body to a non-conforming deformable mesh. This means that the nodes of the rigid body do not necessarily coincide with the nodes of the deformable mesh. This is convenient since this allows you to mesh the two objects separately without being concerned about continuity at the interface.

2.3.5 Step 5: Defining the analysis step

Finally, an analysis step needs to be defined. Select the *Physics/Add Step* menu. Select *Structural Mechanics*. Click *OK* to add the step to the model. This completes the tutorial.

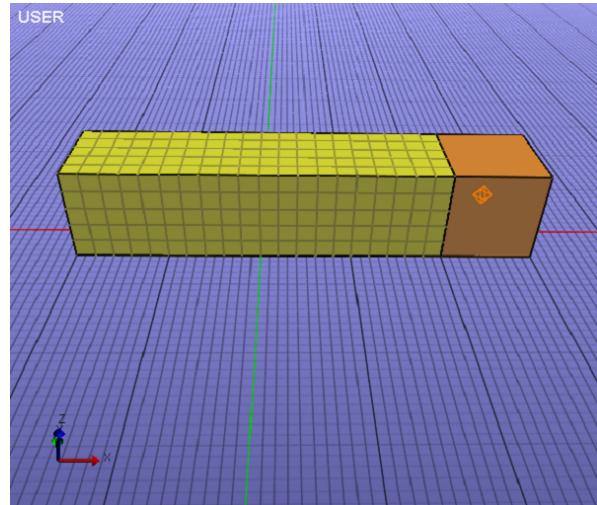


Figure 2.7: Completed model for tutorial 3.

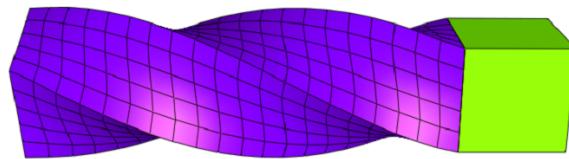


Figure 2.8: The final twisted box.

2.4 Tutorial 4: The billet problem

In this chapter we will create a finite element input file for a compression test where a rigid wall compresses a deformable mesh. The contacting surfaces are allowed to slide across each other without friction. The geometry of the problem is depicted in Figure 9. Due to the symmetry of the problem we will only model the lower right quadrant. This also requires that we set up symmetry boundary conditions. In addition, we assume plane strain conditions which will require only one layer of hex elements through the thickness.

2.4.1 Step 1: Creating the geometry

The geometry consists of a rectangular box. Due to symmetry in the model and boundary conditions, only one quadrant of the box needs to be modeled. First, create the box by switching to the *Create* panel, clicking the *Box* button and entering the following creation parameters:

- width = 2
- height = 2
- depth = 0.1

Click on the *Create* button to create the box. Next, activate the *Mesh* panel and enter the following meshing parameters:

- Mesh type = Butterfly 2D
- Nx = 20

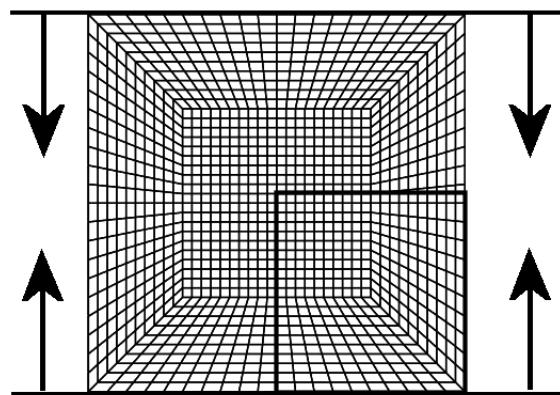


Figure 2.9: The geometry of the billet problem. Due to the symmetry in this problem, we will only model one quadrant.

- $N_y = 20$
- $N_z = 1$
- Segments = 10

Leave the other parameters to their default values and click the *Apply* button. Remember, to view the mesh click the “*m*” shortcut button or select the *View/Toggle Mesh lines* menu from the menu bar.

In order to delete part of the mesh we first need to convert the object to an *Editable Mesh*. With the object selected, click the *convert* button on the Object pane at the top of the create panel and select the *Editable Mesh* option from the dropdown menu. What has happened is that by modifying the object to an editable mesh, PreView gives the user direct access to the FE mesh. You can now edit the mesh directly at the element and node level. On the toolbar at the bottom of the Graphics View you will now see additional buttons that allow you to select mesh items (i.e. elements, faces, edge, and nodes).

On the Graphics View control bar, click on the element button (i.e. the button with the red cube). Select all elements but the ones in the lower right quadrant (note that an easyway to do this is to select the lower right quadrant and then select the menu *Edit/Invert Selection*). Your model should look something like Figure 2.10.

Now delete the selected elements by selecting the menu *Edit/Delete Selection* or by clicking the *delete* button on your keyboard. We are left with the lower right quadrant. Now leave the sub-object selection mode by selecting the element selection button again or by pressing the Esc button.

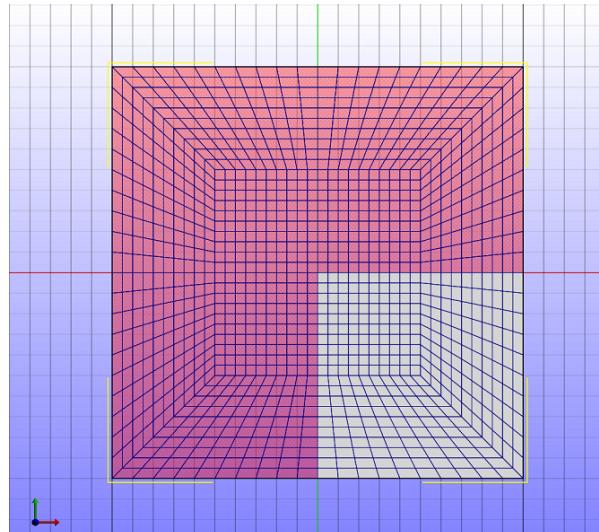


Figure 2.10: The billet mesh with the elements selected which are about to be deleted.

2.4.2 Step 2: Setting up the boundary conditions

First, we'll set up the plane strain boundary condition. This boundary condition implies that no nodes can move out of the plane. We can enforce this by selecting the front and back face of the box and constraining it in the z-direction. To select the faces, first select the *Select Faces* button 

on the main toolbar. Next, select both the top (+z) and bottom (-z) faces of the mesh. Then, apply a fixed constraint by selecting the *Physics/Add Boundary Condition* and selecting *Fixed constraint* from the list. In the properties list, check the *z-displacement* degree of freedom.

Next, we need to set up the symmetry boundary conditions. Basically, we need to constrain the back face (+y) and the left face (-x) so that the nodes on these faces can only move in the respective planes and not normal to the planes. Starting with the left face, select it and apply a fixed constraint to the *x*degree of freedom of this face. Deselect the selected face by pressing the ESC button or by clicking somewhere outside your mesh. Now, select the back face (+y) and constrain it in the *y*-direction. Finally, we will also constrain the front face (-y) so that it can only move in the *y*-direction. Select the face and apply a fixed constraint on the *x*degree of freedom (the *z*-degree of freedom was already as a result of the first boundary condition).

Next, we will create the contact interface. Go to the *Contact Editor* using the *Physics/Add Contact* menu. Select *rigid wall* and click on the *OK* button and. The rigid wall interface creates a contact interface between your mesh and a rigid wall. The rigid wall is an implicitly defined surface so it does not need to be modeled and we do not need to specify any geometry for it. Instead it is described using the plane equation.

$$ax + by + cz + d = 0$$

The four plane coefficients are set in the properties list of the rigid wall item. Set the parameters as follows:

- augmented Lagrangian = Yes
- augmentation tolerance = 0.01
- penalty factor = 1000
- a = 0
- b = 1
- c = 0
- d = -1
- plane displacement = 1

This places the rigid wall at the bottom of the mesh, moving upwards. The rigid wall will now be visible in the Graphics View.

Next, we need to define which surface of the mesh will be in contact with the rigid wall. To do this, click on the “Select Face” button on the main toolbar to activate face selection. Now select the front (-y) and right (+x) faces. Then, click on the rigid wall interface in the Model Editor and find the “Selection” box in the properties window below the Model Editor. Add the currently selected faces to the selection by clicking on the “+” button. This completes the definition of the rigid wall interface.

Next, we will prescribe the motion of the rigid wall. Open the *Curve Editor* from the *Tools/Curve Editor* menu or select the corresponding toolbar button. The Curve Editor displays all the time-dependent parameters in the model and allows you to view and edit the corresponding curves easily (Figure 2.11).

The left hand side shows all the parameters that are time-dependant. Find and select the *displacement* degree of freedom of the rigid wall interface in the Contact section. In the curve view – the right part of the Curve Editor showing the actual curve – select the rightmost point.

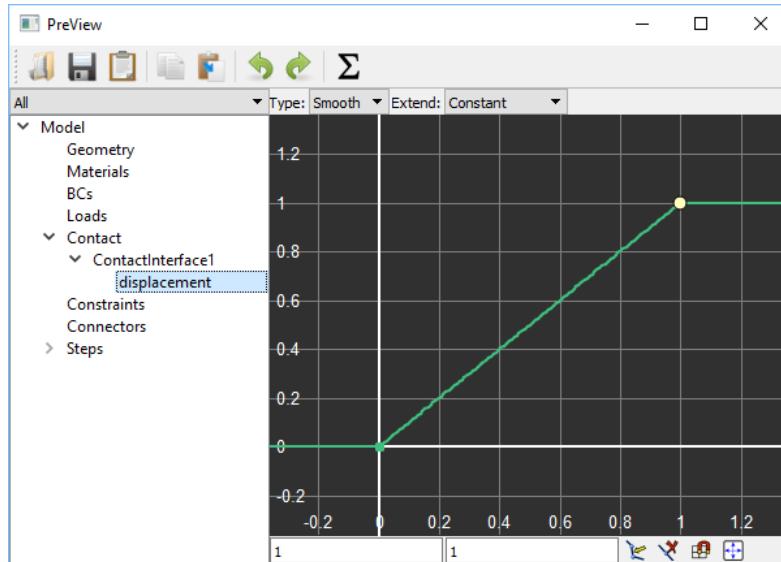


Figure 2.11: The Curve Editor showing the rigid wall's displacement curve.

Once selected, it should turn white. On the bottom of the curve view the coordinates of this point are given. Change them to $(1, 0.4)$. Exit the curve editor. This completes all boundary conditions.

2.4.3 Step 3: Setting up the materials

By now, you should be familiar with setting up materials. Add a neo-Hookean material and give it the following material parameters.

- Young's modulus = 1.0
- Poisson's ratio = 0.49

Now, assign this material to the box.

2.4.4 Step 4: Setting up the analysis

As in the previous tutorials, add an analysis step to the model, accepting the default settings. This completes this tutorial. The final deformed state is displayed in Figure 2.12.

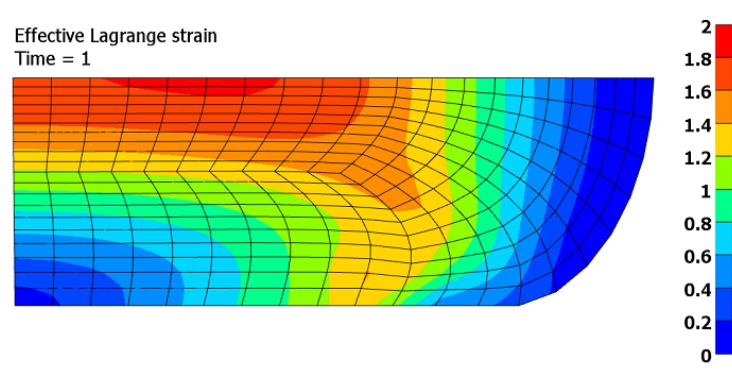


Figure 2.12: The final deformed state of the billet problem showing the effective strain.

2.5 Tutorial 5: Biphasic unconfined compression

In this tutorial we model the unconfined compression of a porous biphasic material. The geometry consists of a cylinder to which a prescribed displacement is applied in the axial direction. After loading the sample, the displacement is held constant while the material relaxes in the radial direction due to the interstitial fluid flow through the material boundaries.

2.5.1 Step 1: Creating the geometry

Create a cylinder with a radius of 1 and a height of 1. To reduce the computational time, we will again create a quarter symmetry problem and setup proper symmetric boundary conditions. This implies that we first need to create the mesh. (Remember to use the *m* shortcut to show the mesh in the Graphics View). Make sure the cylinder is selected and activate the Mesh panel. Set the meshing parameters as follows:

- Meshing Type = Wedged center
- divisions = 30
- segments = 30
- stacks = 1
- R-bias = 0.9

Leave the other parameters unchanged. Press the Apply button to change the mesh. Next, on the *Object* panel click the *convert* button to create an editable mesh. Converting the mesh to an editable mesh allows us to modify the mesh directly on the element level.

On the control bar at the bottom of the Graphics View click on the *Element* sub-object selection button. We now wish to select one quarter of the mesh. To simplify the selection it is easier to work in the top-view. PreView defines several predefined work-planes. To activate the top work-plane, right-click anywhere in the Graphics View and select *Top* from the popup menu. Proceed with selecting all the elements but the upper right corner. Remember that you can select multiple elements by holding down the shift button while dragging with the left mouse button down across the screen.

If your screen currently looks similar to Figure 2.13, you may go ahead and delete all the selected elements by pressing the delete button on your keyboard or by selecting the *Edit/Delete Selection* menu.

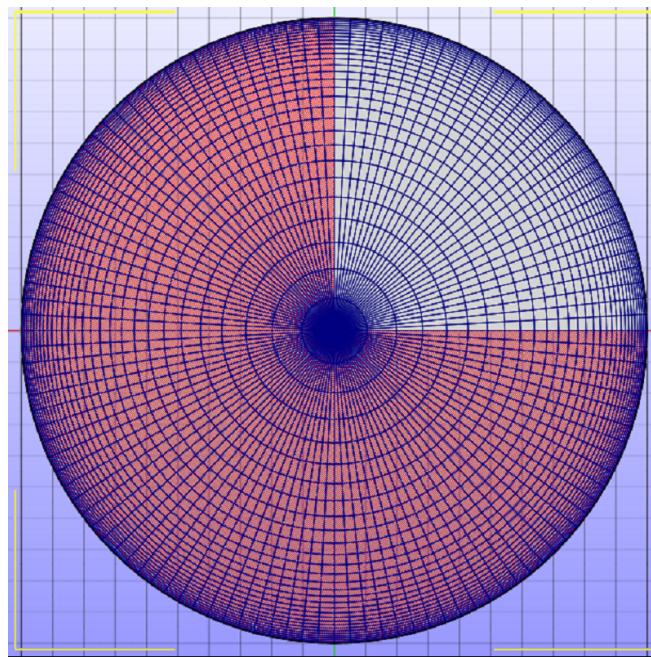


Figure 2.13: The cylinder with all elements selected but the upper right corner.

2.5.2 Step 2: Setting up the boundary conditions

First, we'll apply the symmetry boundary conditions to ensure that the quarter symmetry model is a correct representation of the full model. If you are still in the sub-object mode, press the Esc button to go back to the object mode. Click the *select face* button on the toolbar . Select the yz-plane of the cylinder and apply a fixed constraint to constrain the x degree of freedom.

Next, select the xz-plane of the cylinder and apply a fixed constraint to constrain the y degree of freedom. Also, apply a fixed constraint to the bottom plane (xy-plane) in the z direction. Finally we add one more fixed constraint, namely on the outer curved surface of the cylinder for which we constrain the p (i.e. fluid pressure) degree of freedom. Select the *Physics/Add Boundary Condition* menu, and then select the *Fixed Fluid Pressure* in the dialog box that shows up. Press OK and the boundary condition will show up in the Model Viewer. Make sure to check the checkbox for p in the properties list. The reason that we constrain the pressure degree of freedom is because we want to make this a free-draining surface. The fluid is allowed to leave through a free-draining surface.

To summarize, you should have four boundary conditions thus far:

- the yz-plane: constrained in x .
- the xz-plane: constrained in y .
- the xy-plane: constrained in z .
- the outer, curved surface: constrained in p .

To define the kinematics of the problem, apply a prescribed displacement to the top plane in the z direction. Leave the scale factor set to 1 since we will define the displacement through a curve. Open the Curve Editor (*Tools/Curve Editor* menu) and locate the curve of the prescribed constraint. Select the first point of the curve and move it to the coordinates (0, -0.01). It's easier to type these numbers in the edit field below the curve than to move the node manually. Move the other node to (2000, -0.01). This point will lay off the current plot area. To zoom out to see

the entire curve, click the *zoom all* button below the curve view . This concludes the boundary conditions.

2.5.3 Step 3: Defining material parameters

Since we want to solve a biphasic problem, we need to create a *biphasic* material. Right-click on the *Materials* item in the Model Viewer and select *Add Material* from the popup-menu. In the Material Browser, select the multiphasic category and then select the *Biphasic* material option. Select an *isotropic elastic* material for the solid part and *perm-const-iso* for the permeability. Then, click OK and the material will be added to the Model Viewer. Note that the components of the material (the solid and permeability components) are children of the material item and can be selected separately. In the corresponding property lists enter the following material parameters:

- *Young's modulus* = 1.0
- *Poisson's ratio* = 0.15
- *permeability* = 0.001.

Leave all the other parameters at their default values. Finally, assign this material to the cylinder.

2.5.4 Step 4: Setting up the analysis

Add a new analysis step to the model using the *Physics/Add Step* from the menu and choose *Biphasic* analysis. In the properties list, set the time settings as follows:

- Time steps = 20000
- Step size = 0.1
- Max step size = 100
- Min step size = 0.001
- Auto time stepper = checked
- Optimal iterations = 40

This concludes the tutorial. The Figure 2.15 shows the fluid pressure for four different time steps.

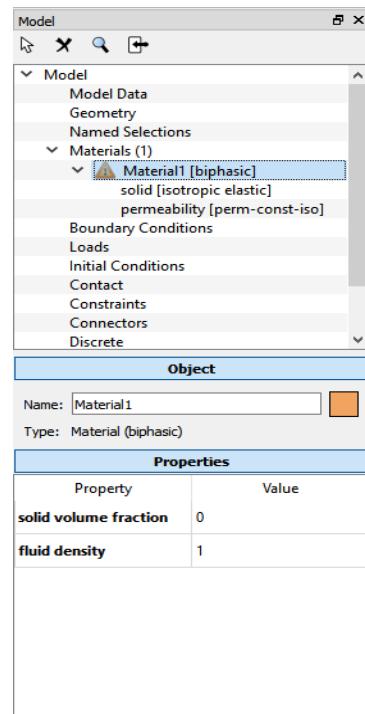


Figure 2.14: The Model Viewer showing the newly created biphasic material and its components.

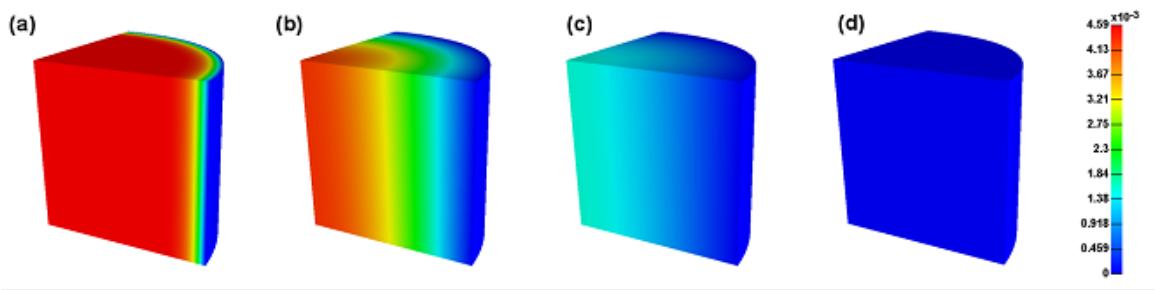


Figure 2.15: Final result for tutorial 5, showing the equilibrium of fluid pressure over time.

2.6 Tutorial 6: A multi-step analysis

In this tutorial we'll explore the multi-step analysis feature. We create a beam bending problem where the beam is loaded statically. Then the load is released and the dynamic response of the beam is explored.

2.6.1 Step 1: Creating the geometry

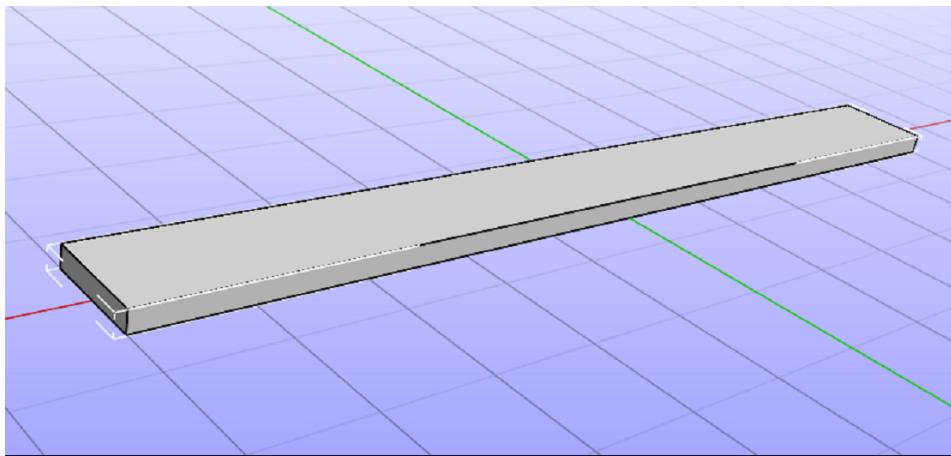


Figure 2.16: The geometry for the multi-step analysis tutorial.

The geometry will consist of a single beam. Activate the *Create* panel and click on the *Box* button. Create a box of width 10, height 1 and depth 0.2.

2.6.2 Step 2: Defining the analysis steps

Before we apply boundary conditions, we need to take a closer look at the *Model Viewer* and in particular at the *Steps* item. You'll see that PreView by default creates a single step, named the *Initial* step. The initial step serves a special purpose since it collects all the constraints, such as boundary conditions, loads, contact interfaces, etc. that do not depend on a particular step. In other words, all the constraints that are defined for this step will be applied during the initialization phase of the analysis and will remain active throughout all the steps. This is different from constraints defined in an analysis step, since those constraints will only remain active during that particular step.

Since we will apply a load only during the static phase, we first create a step to model this phase. Add a *Structural Mechanics* analysis step as was done before in the previous tutorials by either selecting the *Physics/Add Step* menu or by right-clicking on the *Steps* item in the Model Viewer. In the *Time Stepping* properties set the *time steps* to 5 and the *time step size* to 0.2. Uncheck the *auto-time stepper* option.

Next, create another *Structural Mechanics* step in a similar way. In the time settings set the *time steps* to 40 and the *time step size* to 1.0. This step will be a dynamic analysis so change the *analysis type* in the *Analysis* settings to dynamic.

To summarize, we now have three steps defined: the initial step where we will place all persistent boundary conditions that remain active during the entire analysis, a first quasi-static analysis step during which we will apply the load, and finally a dynamic analysis step to simulate the free vibration response of the beam.

2.6.3 Step 3: Setting up the boundary conditions

With all the steps defined, we can now go ahead with creating the boundary conditions. The first boundary condition will be a fixed constraint on the left end of the beam. Click the *Select Face* button on the toolbar  and select the left face of the beam, i.e. the plane parallel to the *yz*-coordinate plane. Select *Physics/Add Boundary Condition*. Notice that at the top of dialog box that appears there is a drop-down list that allows the user to select the step to which this boundary condition needs to be applied. Since we will apply a fixed constraint that must remain active during all steps, select the *Initial* step from the list. Note that this is the default so you should not have to explicitly select it. Now select the *Fixed Constraint* from the list and click *OK*. In the properties list, select the *x, y, and z displacement* degrees of freedom.

Notice the changes in the Model Editor. The fixed displacement we just applied has shown up in the *Boundary Conditions* item as well as in the *Initial* step item. The *Boundary Conditions* item collects all the boundary conditions that are applied to the model, whereas the *Initial* step will only list the boundary conditions that were created in this step (Figure 2.17).

Next, we will apply the load on the other end of the beam. Select the right face of the beam, making sure that the left face is no longer selected. Select *Physics/Add Surface Load* from the menu. In the dialog box that appears select *Step1* from the drop-down list since we want this load only to be active during the static phase of the analysis. Select *Surface traction* load from the list and click *OK*. The traction allows us to apply a constant traction to a surface, that is, a constant force per unit area. This is often more convenient than applying a nodal load on a surface since in the latter case the total effective force will depend on the number of nodes on that surface. The traction load remains independent of the mesh. In the properties list, enter the value:

- traction = 0, 0, -1

Notice that the traction load has shown up in the *Loads* item and in the *Step1* item of the Model Editor.

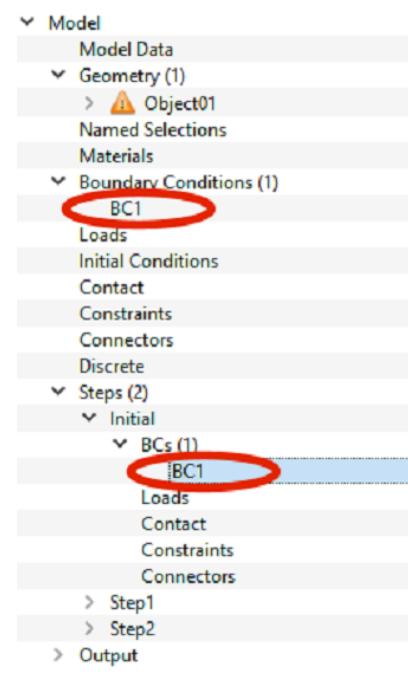


Figure 2.17: The BC1 boundary condition is now visible in the overall boundary condition list and under the initial step.

2.6.4 Step 4: Defining material parameters

Create a new material by selecting the *Physics/Add Material* from the menu. Select the *isotropic elastic* material and click Add. Enter the following material parameters:

- Young's modulus = 25000
- Poisson's ratio = 0.35.

Assign this material to the beam.

2.6.5 Step 5: Creating the mesh

Only one task remains, and that is the creation of the mesh. With the beam selected, activate the *Mesh* panel. Enter the following meshing parameters.

- Nz = 20
- Ny = 5
- Nz = 3
- X-bias = 1.2

Leave the other parameters at their default values. The *X-bias* parameter allows you to refine the mesh at one end. Since we anticipate that the beam stresses will be higher at the fixed end, we want to create a more accurate solution by increasing the mesh density at this end. Click *Apply* to create the new mesh and to complete this tutorial.

2.7 Tutorial 7: Heat Transfer Analysis

In this tutorial we will setup a steady-state heat transfer problem. The particular problem considered here is a standard test problem for heat-transfer codes and an analytical solution is available for verification.

2.7.1 Step 1: Geometry and Meshing

The geometry consists of a rectangular box with 2:1 ratio for the size. To create this box, open the Create panel, select the box, and enter the following dimensions in the appropriate fields.

- Width = 0.1
- Height = 0.05
- Depth = 0.01

Click the *Create* button to create the geometry. The box, which is named *Object01* by default, will now show up in the Model Viewer, on the left side of the screen.

To create a mesh for this box, activate the Mesh panel and enter the following meshing parameters:

- Nx = 20
- Ny = 10
- Nz = 1

Leave the other parameters at their default values and click *Apply*. This will create a regular hexahedral mesh for the box.

2.7.2 Step 2: Choosing a material

Open the material browser. This can be done from the menu Physics/Add Material, or by right-clicking on the Materials item in the Model Viewer and selecting the Add Material menu. The Material Browser appears with a list of supported materials. Select the heat transfer category and then select the Isotropic Fourier material. Click the Add button. The material, by default named *Material1*, now shows up in the Materials items of the Model Viewer. To assign the material to the box, first select the box and click the plus sign in the material's selection box. Enter the following material properties.

- Density = 1 (kg/m^3)
- Heat conductivity = 0.4 ($\text{W}/(\text{m}\cdot\text{°C})$)
- Heat capacity = 1 ($\text{J}/(\text{kg}\cdot\text{°C})$)

Note that in this particular problem only the conductivity plays a role since we will be modeling a steady-state problem.

2.7.3 Step 3: Boundary Conditions

The following boundary conditions will be applied to the box. The left face of the box will have a fixed heat flux prescribed, the right face of the box will be held at a fixed temperature, the bottom face will be considered adiabatic, and the top face will have a convective heat transfer applied.

Let's begin with the left face. Activate the face-selection mode by clicking the appropriate button in the toolbar (button with blue face). Select the left face of the box. From the menu, select Physics/Add Surface Load and from the list of loads that shows up, select the "heat flux" option. Click OK and in the next dialog box enter 3,500 (W/m²) for the flux property.

Now select the right face (make sure the left face is no longer selected) and select the Physics/Add Boundary Condition menu. From the list, select the Prescribed Temperature and click OK. In the properties list, enter 25 (°C) for the scale factor.

Next, select the top face and select the Physics/Add Surface Load menu. This time select the "convective heat flux" option and click OK. In the next dialog box, enter 60 (W/m²·°C) for the heat transfer coefficient and 25 (°C) for the ambient temperature.

Finally, we will also add a heat source to this model. The heat source will be applied to the entire model, so no selection is needed. From the menu, select Physics/Add Body Load and select heat source. Enter 1.353×10^5 W/m³ for the Q parameter. This completes the model setup.

2.7.4 Step 4: Defining a heat transfer analysis

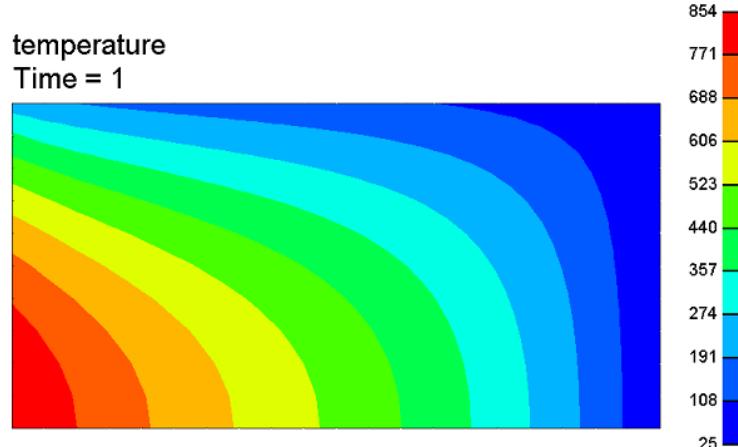


Figure 2.18: Temperature contours for the problem described in Tutorial 7.

The last thing that remains is to setup the analysis. To do this, select Physics/Add Step from the menu and select the Heat transfer option from the list. Click OK. Leave all parameters with their default values, but check-out the analysis option. The default analysis is set to steady-state, which is what we want. However, if you want to run a transient analysis, this is where you can make the appropriate change. This completes the analysis setup. The result is shown in Figure 2.18.

2.8 Tutorial 8: Reversible binding kinetics

The objective of this tutorial is to model reversible binding kinetics described by the reaction



where R denotes the receptor, L is the ligand and C is the receptor-ligand complex. Consider that the receptor and complex are solid-bound molecules whereas the ligand is a solute in a multiphasic mixture representing a biological tissue. For example, the ligand may represent insulin-like growth factor I (IGF-I) and the receptor may represent the IGF-I binding protein 6 (IGFBP-6). The receptor and complex may be bound to the extracellular matrix of articular cartilage.

2.8.1 Step 1: Defining the model data

Multiphasic materials require the specification of global variables. In the Model Viewer select Model Data and enter the following values:

- Absolute temperature: 293 K
- Gas constant: 8.314 nJ/nmol·K
- Faraday's constant: 96.485 μ C/nmol

2.8.2 Step 1: Defining the solute and solid-bound molecules

Select *Physics/Solute Table* from the menu. In the *Solute Table* dialog box click on *Add*. A new entry will be added to the table with some default values. You can double-click on any value to change it. Change the values to the following:

- *Name*: ligand
- *Molar Mass*: 7.649×10^{-6} g/nmol
- *Density*: 1×10^{-3} g/mm³

Then close the *Solute Table* dialog window. Select *Physics/Solid-Bound Molecule Table* from the menu. Click *Add* and enter the following values:

- *Name*: receptor
- *Molar Mass*: 23×10^{-6} g/nmol
- *Density*: 1×10^{-3} g/mm³

Add one more solid-bound molecule with the following values:

- *Name*: complex
- *Molar Mass*: 30.649×10^{-6} g/nmol
- *Density*: 1×10^{-3} g/mm³

Then close the dialog window.

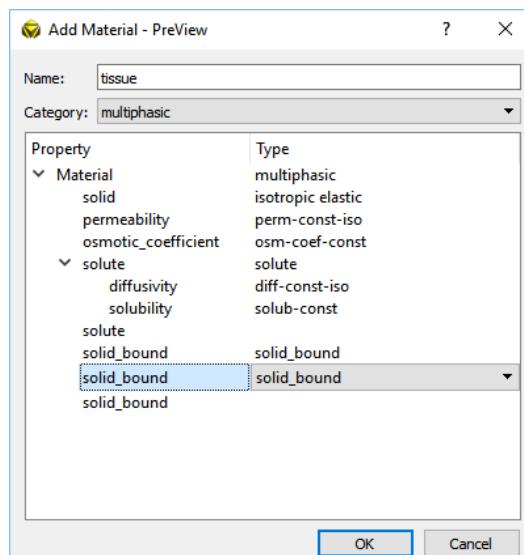


Figure 2.19: The multiphasic material's composition for this tutorial.

2.8.3 Step 2: Creating a multiphasic material

Since we want to solve a multiphasic problem, we need to create a *multiphasic* material. Select and right-click on the *Materials* item in the Model Viewer and select *Add Material* from the popup-menu. Rename the material *tissue* in the *Name* entry. Select *multiphasic* as the category. Find the *Multiphasic* material option in the material item. Select an *isotropic elastic* material for the solid part, *constant* for the permeability and *constant* for the osmotic coefficient. Add a solute and select *constant* diffusivity, and *constant* solubility. Then add two solid-bound components. The material's composition should now look like in Figure 19. Click Ok to exit the dialog window.

This process only defines the composition of the multiphasic material and adds it to your model. We still need to set the properties of all the material components. The components of the material can be selected in the Model Viewer by expanding the *tissue* item. In the property list, below the Model Viewer, enter the following material parameters:

- *tissue*:
 - *Solid volume fraction*: 0.2
- *solid*:
 - *Young's modulus*: 10^6 Pa
- *permeability*:
 - *permeability*: 10^{-9} mm⁴/μN·s
- *solute*:
 - *solute: ligand*
 - *diffusivity*:
 - * *free diffusivity*: 5×10^{-3} mm²/s
 - * *diffusivity*: 4×10^{-3} mm²/s
- *solidbound (1)*
 - *Solid-bound molecule*: *receptor*
 - *apparent density*: 0.92×10^{-9} g/mm³
- *solidbound (2)*
 - *Solid-bound molecule*: *complex*

All other parameters should be left at their default values.

2.8.4 Step 3: Creating a chemical reaction

Select *Physics/Add Chemical Reaction* from the menu. In the *Add chemical reaction* dialog box enter *Reversible Binding* as the *Name*; pull down the *Multiphasic* menu and select *Tissue*; pull down the *Reaction* menu and select *mass-action-reversible*; pull down the *Fwd Rate* menu and select *constant reaction rate*; pull down the *Rev Rate* menu and select *constant reaction rate*. Under *Reactants* you'll see two panels. The left panel shows all available reactants and the right panel shows the actual reactants that will be used in this reaction. Now select the ligand and

receptor and move them from the left panel to the right panel. The *Products* panels work similarly. Move the *complex* product form the left to the right panel. Click Ok to exit this dialog window.

This process adds a chemical reaction to the *tissue* material in your model. Select *tissue* in the Model Viewer and expand it until the reaction components are visible. Then enter the following material parameters:

- *forwardrate k*: $200 \text{ mM}^{-1} \cdot \text{s}^{-1}$
- *reverserate k*: 10^{-3} s^{-1}
- *ligand vR*: 1
- *receptor vR*: 1
- *complex vP*: 1

2.8.5 Step 4: Geometry and meshing

The geometry consists of a cube. Open the Create panel, select the box, and accept the following default dimensions in the appropriate editing fields.

- Width = 1 mm
- Height = 1mm
- Depth = 1 mm

Click the *Create* button to create the geometry. The box, which is named *Object01* by default, will now show up in the Model Viewer.

To create a mesh for this box, activate the Mesh panel and enter the following meshing parameters:

- Nx = 1
- Ny = 1
- Nz = 30
- z-Bias = 0.9

Leave the other parameters with their default values and click *Apply*. This will create a biased hexahedral mesh for the box. Remember to use the ‘m’ shortcut to toggle the mesh in the Graphics View.

Assign the multiphasic material *Tissue* to this box by first selecting the box then clicking the plus sign in the material’s selection box.

2.8.6 Step 5: Boundary and initial conditions

Use *Physics/Add Boundary Condition* (or Ctrl-B) to prescribe boundary conditions. Select the two faces normal to the *x*-direction and constrain them to have fixed displacement along *x*. Similarly, select the two faces normal to the *y*-direction and constrain them to have fixed displacement along *y*. Select the *z = 0* face normal to the *z*-direction and constrain it to have a fixed displacement along *z*.

Select the $z = 1$ face normal to the z -direction and prescribe the effective solute concentration for the ligand and the effective fluid pressure. Keep the default value of 1 for the scale factor in both cases. Use View/Curve Editor (or F4) to call up the curve editor and select the *Prescribed-Concentration BC* for the ligand; switch the curve type to *Step*, select the second point (initially at 1,1) and enter the ordinate values 10^{-5} mM. Similarly, select the *PrescribedFluidPressure BC*, switch the curve type to *Step*, select the second point and enter the ordinate value -2.436×10^{-2} Pa (evaluated from the negated product of the gas constant, the absolute temperature and the prescribed ligand concentration). Close the curve editor.

The initial conditions for the ligand concentration and fluid pressure in this problem are zero, which is the default value for all initial conditions in PreView. Therefore it is not necessary to explicitly set the initial conditions here. For completeness however, you may wish to go through the following steps. In the main toolbar click on the *Select Part* icon and select the box. Select *Physics/Add Initial Condition* (or Ctrl-I), then select *Initial Concentration* in the dialog and click Ok. Select the initial condition in the Model Viewer and in the properties list select the *ligand* solute and set the *value* to 0.

2.8.7 Step 6: Setting up the analysis

Add a new analysis step to the model using the *Physics/Add Step* from the menu and choose *Multiphasic* analysis. Select the *Time Stepping* settings as follows:

- *Time steps* = 10000
- *Step size* = 1
- *Max step size* = 1000
- *Min step size* = 0.1
- *Auto time stepper* =checked
- *Use must points* =checked

Leave other parameters at their default value.

Set the *Nonlinear solver* settings to:

- *Displacement tolerance* = 0
- *Energy tolerance* = 0
- *Pressure tolerance* = 0
- *Concentration tolerance* = 0.01
- *Max updates* = 0

Set the *Matrix Storage* in the *Linear solver* settings to *Non-symmetric*.

Bring up the Curve Editor (F4) and edit the *must point* load curve in *Step01* to have a *Linear* curve type, with the second point located at (10000, 2000). This setting will increase the maximum time step linearly from 0 to 2000 s over the span of the analysis time.

In the Model Viewer select the *Output>Plotfile* and right-click on it to *Edit output* these options. Set the default output variables and also click on *sbm 1 concentration (receptor)* and *sbm 2 concentration (ligand)*.

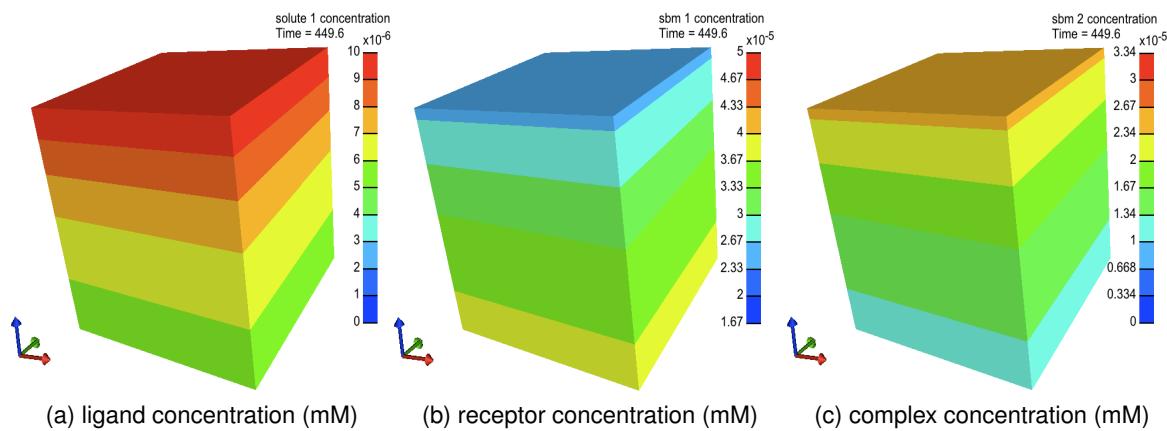


Figure 2.20: Reversible binding kinetics

This concludes the tutorial. Figure 2.20a below shows the *solute 1 (ligand)* concentration, the *sbm 1 (receptor)* concentration, and the *sbm 2 (complex)* concentration at a select intermediate time step.

2.9 Tutorial 9: Axisymmetric analysis of biphasic indentation

This tutorial examines the indentation of a biphasic layer with a rigid impermeable frictionless indenter under creep loading. The purpose of this tutorial is to demonstrate the 3D modeling of axisymmetric problems using wedge geometries and a plane of symmetry not aligned with the coordinate planes (Figure 2.21). The tutorial also shows how to create biased meshes, and attaching and welding congruent meshes.

2.9.1 Creating the geometry

Since FEBio does not provide 2D axisymmetric elements, an axisymmetric analysis may be simulated in 3D using only a wedge (slice) out of the true 3D geometry. Reducing the wedge angle increases the faithfulness of the 3D analysis to the 2D axisymmetric analysis. In this tutorial we will use a wedge angle of 3 degrees. The first objective is to create the wedge representing the biphasic layer and mesh it as shown in Figure 2.22.

Note that the mesh is biased along the radial and axial directions. The axial direction has a dual bias (the mesh is finer at both ends and coarser in the middle). This type of biasing is a standard feature that can be implemented by simply checking a box and entering a bias value as shown below. The radial direction also shows a dual bias, but the location of the dividing line is not at the midpoint of the radial extent. To reproduce this non-standard bias, two geometries must be created, meshed congruently, then attached and welded.

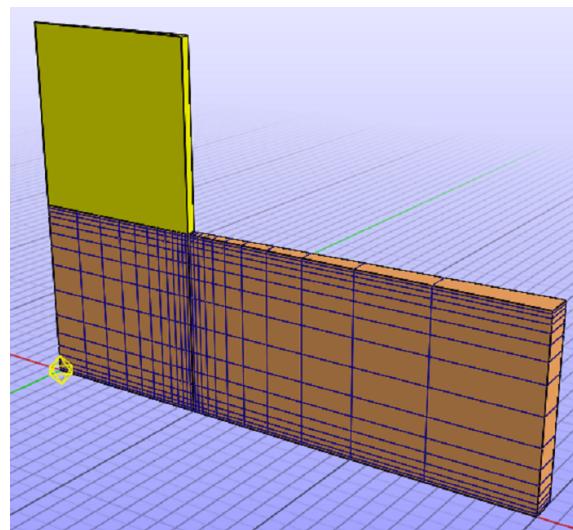


Figure 2.21: Model for axisymmetric analysis of biphasic indentation.

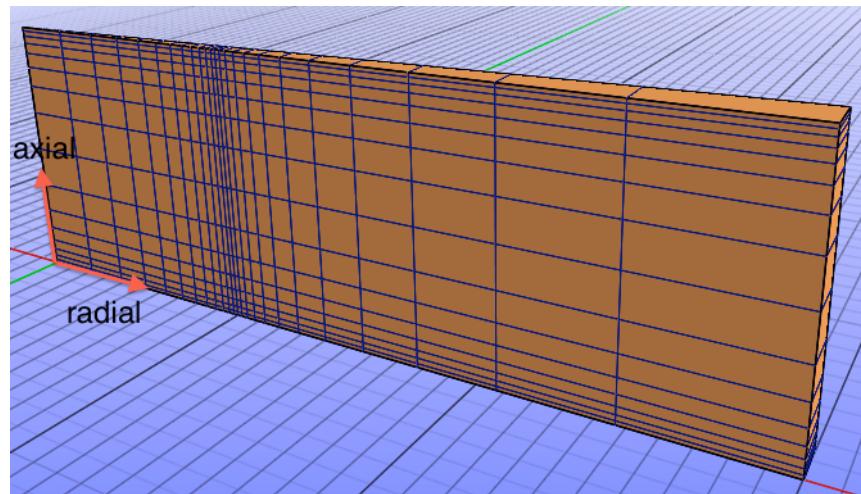


Figure 2.22: Geometry and biased mesh for tissue layer in indentation analysis. In the radial direction, the mesh is finest at the location coinciding with the indenter edge.

First, create the inner *Slice*  with Radius=1, Height=1, Angle (deg)=3. Second,

create a *Tube*  with Inner Radius=1, Outer Radius=3, Height=1. Mesh the slice using Slices=1, Segments=12, Stacks=16, Z-bias=1.4, R-bias=0.8, Z-mirrored bias=checked. Mesh the tube to be congruent with the slice along the axial and circumferential directions: Slices=30 (to get 3 degrees per slice), Segments=12, Stacks=16, z-bias=1.4, r-bias=1.4, z-mirrored bias = checked. The mesh should now look as shown in Figure 2.23.

Convert both geometries to *editable meshes*. For the outer tube, delete all elements except the outer slice that meets the inner slice (Figure 2.24). Next, we'll merge the two objects into a single object. To do this, select both objects. Then, select the menu *Edit\Merge objects*. In the dialog box that appears make sure the *weld* option is checked and press OK. This completes the geometry and mesh for the tissue layer

To create the indenter, activate the Create panel again and create another slice with Radius=1, Height=1, Angle (deg)=3, at the position x=0, y=0, z=1. Mesh it with Slices=1, Segments=1, Stacks=1 (Figure 2.25). For a creep analysis, where a load is prescribed on the indenter, the contact between the indenter and tissue layer will fail unless the two contacting surfaces overlap at the initial time point. Therefore, a small initial overlap should be created between the indenter and the tissue layer. Make sure the indenter is selected and select the menu *Edit\Transform*. In the dialog box enter the value -1e-4 in the Relative Z field and press OK.

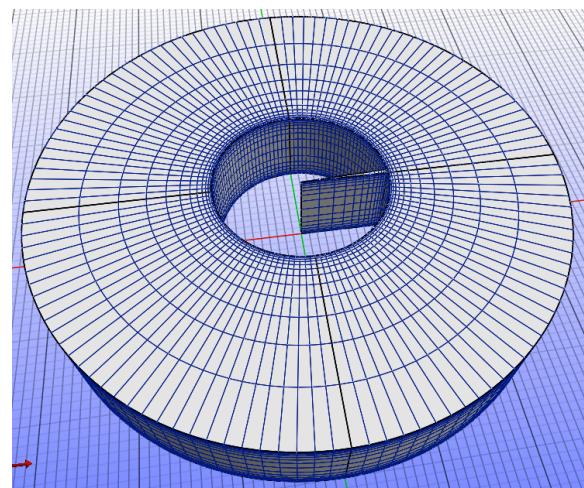


Figure 2.23: Two geometries (slice and tube) serve as building blocks for the final geometry of the tissue layer.

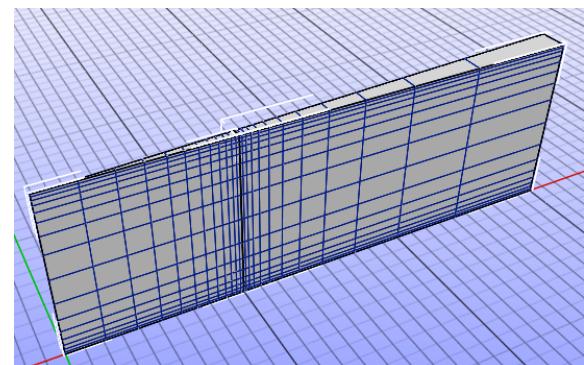


Figure 2.24: Delete all elements in the tube building block, except for the outer slice that meets the inner slice.

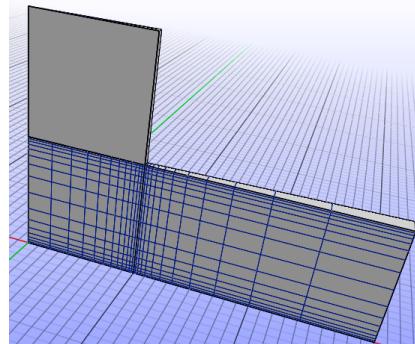


Figure 2.25: Create the indenter using a slice geometry and mesh it with a single element.

2.9.2 Setting up the materials

Create a Biphasic material (call it Tissue) with a neo-Hookean solid and constant permeability. Assign this material to the tissue layer. Set the properties as follows:

- solid volume fraction = 0.2
- Young's modulus = 0.4
- Poisson's ratio = 0.02
- permeability = 0.02

Create a Rigid Body (call it Indenter) and assign it to the indenter.

2.9.3 Setting up the boundary conditions

To enforce an axisymmetric response, a symmetry plane should be created which encompasses the deformable regions. Select the surfaces of the biphasic material which lie on the rotated wedge faces. Then select *Physics/Add Contact/Symmetry* and set

- augmented Lagrangian = No
- penalty factor = 1e6

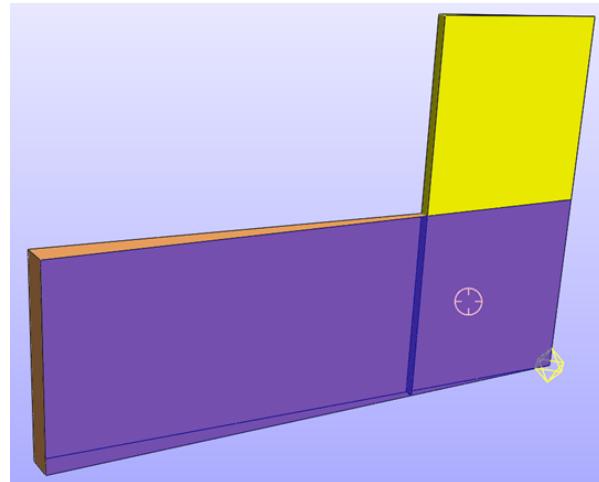


Figure 2.26: Create a symmetry plane on the rotated surfaces of the wedge, using Add Contact/Symmetry plane.

Fix the displacement of its front faces along y. Also fix the x and y displacements of the edge coinciding with the axis symmetry. Fix the x, y and z components of the bottom face of the tissue layer.

Add a Rigid Constraint to the Indenter, with Fixed Displacement/Rotation enforced for x, y, Rx, Ry, Rz, leaving z free.

Add a Rigid Constraint to the Indenter, with *Prescribed rigid force* enforced for z and value of 1.0. Set the load curve for this force to Curve Type = Step with a value of -0.002 at t=1. Note that

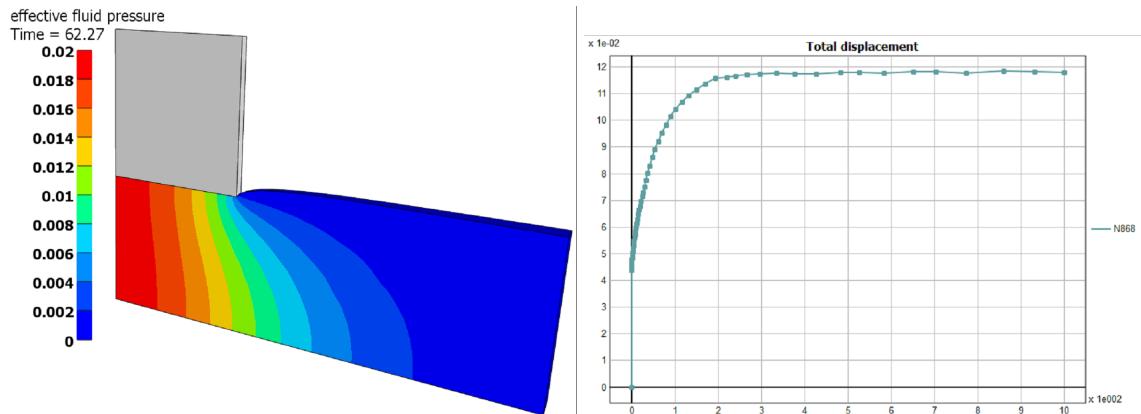


Figure 2.27: Indenter creep displacement (top) and fluid pressure within the tissue layer at an intermediate time point.

the actual force is 120 times this value, since we are using a 3-degree wedge ($360/3=120$) in this axisymmetric analysis.

Add a Biphasic Contact interface between the bottom surface of the Indentor (Master) and the top surface (inner and outer slices) of the Tissue (Slave). You may need to hide the Indenter to select the Tissue contact surface underneath it, and vice-versa to select the Indenter contact surface. Use the following non-default parameter:

- augmented Lagrangian = Yes
- auto-penalty = Yes
- penalty = 1

2.9.4 Defining the analysis step

Add a Biphasic step and use the following non-default parameters:

- Time Stepping
 - Use must points
 - Time steps = 10000
- Nonlinear Solver
 - Max updates = 0
- Linear Solver
 - Matrix storage = Non-symmetric

Edit the loadcurve for must points to be of Curve Type=Linear and include the points (0,0) and (1000,200).

Export the file to Febio format and run the analysis. The resulting indenter creep displacement curve and fluid pressure distribution are shown in Figure 2.27a.

2.10 Tutorial 10: Setting up a model from imported geometry

PreView supports different mesh and surface file formats, including ABAQUS, NIKE3D, ANSYS, STL, BYU, and many more. (See Appendix A for a detailed list of supported formats.) Since PreView doesn't have a lot of mesh generation capabilities, users often create their finite element mesh in another software package and then import it into PreView. From here, boundary conditions and material properties are defined. Although many of the techniques from previous tutorials can be applied directly to imported meshes, some important differences exist between PreView's primitives and imported meshes. This tutorial will walk the reader through importing a mesh and setting up a simple finite element analysis on the imported mesh.

2.10.1 Step 1: Importing the mesh

The first step is to load the mesh from file. To do this, click on the File menu and notice that there are three menu items for loading a file.

- *File\Open*: This opens a *prv* file, PreView's native file format.
- *File\Import FE Model*: This allows users to load a complete FE model, with mesh, boundary conditions, loads, materials, etc. This menu only supports file formats that define complete FE models (e.g. *.feb*)
- *File\Import Geometry*: This allows users to add meshes to their current project.

In this case we want to add some geometry so go to the menu *File\Import Geometry*. This opens a standard file open dialog box. In the file filter list, locate the filter that corresponds to the file format you want to open. In this tutorial we'll use an Abaqus (.inp) file, so select the corresponding filter. Then locate the *box.inp* file and press the Open button. A second dialog box will now open where some options can be set for reading Abaqus files. For our problem the default settings are sufficient so just click OK. PreView will now read the file and show it in the Graphics View. In the Model Viewer you will also see that PreView added an item in the Geometry list with the same name as the imported file.

When PreView reads a mesh, it creates a geometry object for that mesh. It does this by automatically partitioning the mesh and its surface. The surface is partitioned using an algorithm that detects the crease edges in the mesh using an angle criterion. In our case, the default partitioning is sufficient, but the reader can consult Chapter 4 for more details on the auto-partitioning algorithm in PreView and on how to customize the partitioning in case the default partitioning is not the desired partitioning.

Next, we will setup the boundary conditions. When working with imported meshes (in PreView they are also referred to as *editable meshes*) there are two ways for setting up boundary conditions. You can assign boundary conditions and apply loads to the surfaces, edges and nodes of the geometry. Alternatively, you can apply them directly to the mesh' face or nodes. Since both methods have their advantages and disadvantages, we will discuss both approaches.

2.10.2 Step 2: Setting up boundary conditions, method 1

In this step we will apply the boundary conditions directly to the items of the mesh. It will be easier to apply boundary conditions when the mesh is visible so turn on the mesh (if it's not on yet) by pressing the '*m*' shortcut key.

Make sure the object is selected. Notice that for an editable mesh several more options appear on the toolbar at the bottom of the Graphics View. These options allow you to make mesh item selections. Click on the *Select faces* button (red square). Notice that bounding box of the selected object turns yellow. This is to indicate that you are selecting mesh items. Select the bottom faces of the box. If the *Select connected* button is checked you only need to select one facet of the bottom surface. PreView will automatically select all faces that are connected to this face. After all faces of the bottom surface are selected, apply a fixed boundary condition to the selection. To do this, select the *Physics\Add Boundary Condition* menu and select the *Fixed Displacement* option for the dialog box. In the properties list on the Model Viewer, select the x , y , and z degrees of freedom. You have now constrained the bottom surface of the box in all directions.

Next, we will apply a point load to the center node of the top surface. On the Graphics View toolbar, press the *Select Nodes* button. Turn off the *Select connected* option and then select the center node of the top surface. (If the *select connected* option is on, all nodes of the top surface would have been selected.) To apply a point load, select the *Physics→Add Nodal Load* menu. In the following dialog box, select the z -force as the variable and set the load scale factor to -0.1. Press OK to finalize the point load.

2.10.3 Step 2: Setting up boundary conditions, method 2

This section describes an alternative method for applying boundary conditions for imported meshes. As mentioned earlier, when PreView imports a mesh it also generates a geometry object and partitions it based on angle-criteria. Working with the geometry is often more convenient, however sometimes it may be necessary to modify the partitioning of the object. In this case, we want to apply a load to the center vertex of the top surface. However, PreView did not create a geometry node for this vertex. You can see this by activating the (geometry) node selection option on the main toolbar. PreView will draw all the geometry nodes (in blue) and you will see that there is no node in the center of the top surface. We will begin by adding this center vertex to the geometry.

Make sure the object is selected and the object selection is active (green ball) on the main toolbar. On the Graphics View toolbar turn off the *Select connected* option and select the center vertex of the top surface. Activate the Mesh panel and find the *Partition* tool. This tool doesn't have any parameters so just press the *Apply* button. If you now activate the (geometry) node selection option on the main toolbar, you will see that the center vertex is now also a geometry node. With the partitioning completed, we can now go on to applying boundary conditions.

Important Note: *Modifying the partitioning of an object can change the numbering of surfaces, edges, and nodes. It is therefore important to always adjust the partitioning before applying any boundary conditions. Modifying the partitioning after applying boundary conditions may invalidate the boundary conditions and cause problems when exporting the model.*

If you already applied the boundary conditions via method 1, you will first need to delete these boundary conditions. You can do this by selecting them in the Model Viewer and selecting the delete button (icon that looks like an X) at the top of the Model Viewer.

Activate the (geometry) surface selection option from the main toolbar (blue face) and select the bottom surface. Select the *Physics\Add Boundary Condition* and select the *Fixed Displacement* option. In the properties list, select the x , y , and z degrees of freedom. You have now constrained the bottom surface of the box in all directions.

Next, we will apply a point load to the center node of the top surface. Activate the node selection option from the main toolbar and select the center node of the top surface. To apply a point load, select the *Physics→Add Nodal Load* menu. In the following dialog box, select the z -force as the variable and set the load scale factor to -0.1. Press OK to finalize the point load.

2.10.4 Step 3: Finishing the model

To complete the model we need to define a material and an analysis step. To add a material, activate the *Model Viewer* and right-click the *Materials* item. Select the *Add Material* menu and select *neo-Hookean* from the material list. This will add the material to the *Materials* section of the *Model Viewer*. Select the material in the *Model Viewer* and change the material properties to:

- Young's Modulus: 10
- Poisson's ratio: 0.3

To assign this material to the box, select the box and press the plus icon in the *Selection* panel of the material's property window.

Finally, we need to define the analysis step. Right-click on the *Steps* item in the *Model Viewer* and select the *Add Step* menu item. Select the *Structural Mechanics* option from the list and click OK. In the next dialog box just accept all default settings and press the OK button.

This completes the definition of the model. The model can now be exported and is ready to run.

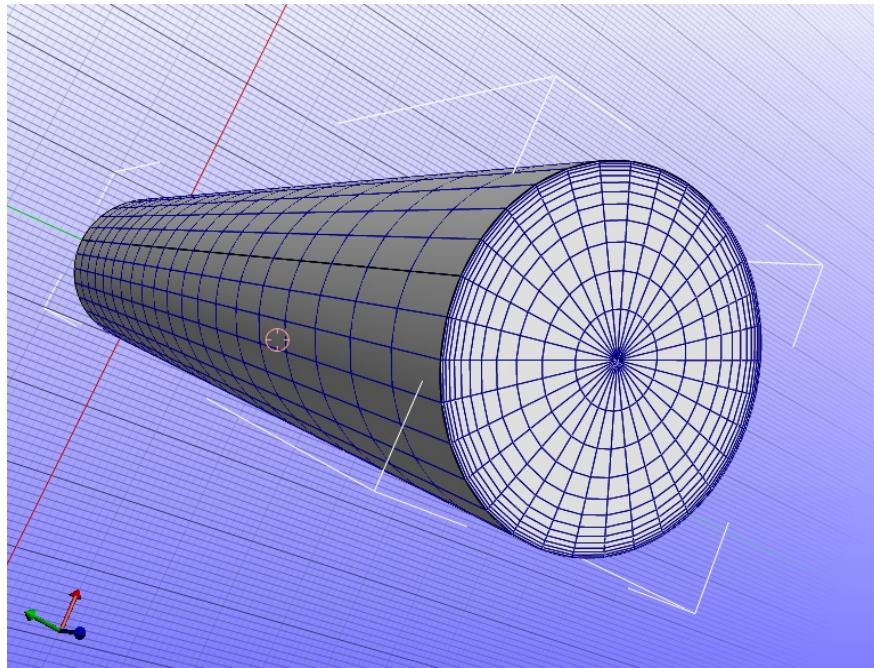


Figure 2.28: The model geometry and mesh for Tutorial 11.

2.11 Tutorial 11: Setting up a Fluid analysis

This tutorial will illustrate how to set up a computational fluid dynamics problem. A simple pipe flow problem will be created where a plug normal fluid flow is prescribed at the inlet and zero pressure is prescribed at the outlet. The flow will be fully developed closer to the outlet of the pipe. Remember to pick the fluid mechanics project template when PreView is opened or when starting a new project.

2.11.1 Step 1: Creating the Geometry and Mesh

The geometry of this problem is a cylinder. Activate the *Create* panel and create a cylinder located at the origin (0,0,0) with parameters *Radius* = 1, *Height*=10. Then activate the *Mesh* panel. Enter the following settings:

- Slices = 8
- Segments = 12
- Stacks = 20
- R-bias = 0.75
- Mesh Type = Wedge center

Leave other parameters at their default values and click the *Apply* button. The model should look as follows in Figure 2.28.

2.11.2 Step 2: Setting up the materials

In this model, we will use a Newtonian fluid, which is the simplest fluids material model as the material parameters do not change and the stress is proportional to the shear rate. It is a good assumption for many common fluids such as water. To add a material to the model, open the *Material Browser* by selecting the *Physics/Add Material* menu. Alternatively you can also open the *Material Browser* by right-clicking on the *Materials* item in the Model Viewer and selecting *Add Material* from the popup menu. There is also a shortcut for this on the toolbar (icon with four colored circles). First, from the Category option select *fluid*. Then, select the *fluid* material and then the *Newtonian fluid* viscous model and click *OK*. The newly added material should now be highlighted in the Model Viewer and the material properties are now displayed in the properties panel, located below the Model Viewer. Then, set the following material properties:

- density = 1.0
- bulk modulus = 1e+09

Like before, with the material selected in the Model Viewer you will notice that it has a selection box that lists the parts to which this material is assigned. After you have selected the object in the Graphics View, click on the plus-sign in the material's selection box, to assign this material to the object. The color of your object will now change to the same color of the material to indicate that it is now associated with that material. Then in the model tree, expand the fluid material and click on *viscous*. Set the following material properties:

- shear viscosity = 0.1
- bulk viscosity = 0

2.11.3 Step 3: Setting up the boundary conditions

Select the four faces on the side of the cylinder and apply a zero fluid velocity boundary condition in the *x*, *y*, and *z* degrees of freedom. This can be done by selecting *Physics* from the menu and then selecting *Add Boundary Condition*. Alternatively, on the model tree you can right click *Boundary Condition* and select *Add Boundary Condition*. Or another way to bring up the boundary condition menu is to select hotkey Ctrl-B (Cmd-B on Mac). Now, in the add BC menu, select *zero fluid velocity* and press *OK*. The boundary condition should be displayed now in the model viewer. Under *Properties* click the check boxes for *x*, *y*, and *z* fluid velocity since we want to fix them. Now, making sure that the “select surfaces” button is clicked on the top, click the four side surfaces, holding Shift key to make it easier, and then click the plus-sign in the BC selection box, so that this BC can be assigned to these surfaces.

Next, we do the same thing as before, except now we are assigning a *Zero Fluid Dilatation*. Make sure the four surfaces previously highlighted are deselected by clicking anywhere in the graphics viewer. In the model viewer under *Properties* check the box for *dilatation*. Assign this boundary condition to the outlet surface, which is the cylinder face NOT on the origin.

Next, we do the same thing in assigning a zero fluid dilatation BC but this time on the boundary of the inlet surface (which contains the origin). Make sure *Select Curves* is selected. Note that this will be four curves. This boundary condition needs to be assigned as the dilatation on the inlet curve is indeterminate as all other degrees of freedom have been assigned.

Finally we have to prescribe the boundary condition for the inlet. This time, select *Physics* from the menu and then select *Add Surface Load*. Alternatively, on the model tree you can right click

Loads and select *Add Surface Load*. Or another way to bring up the boundary condition menu is to select hotkey Ctrl-L (Cmd-L on Mac). Now, from the *Add Surface Load* menu, click on the option *fluid normal velocity* and press *OK*. The surface load should show up in the model viewer. Under *Properties*, we want to set the *velocity*=-1 as the fluid is to flow into the cylinder. Then keeping all the other parameters at their default value, assign this boundary load to the inlet surface, which is the cylinder face on the origin.

2.11.4 Step 4: Setting up the analysis step

Finally, an analysis step needs to be defined. Select the *Physics/Add Analysis Step* menu. Select *Fluid Mechanics*. Click *OK* to add the step to the model. Then change the following parameters under the Model Viewer while keeping all other parameters the same:

- Time steps = 25
- Matrix symmetry = non-symmetric
- Residual tolerance = 0.001
- Max Reformations = 3
- Max updates = 50
- Reform on diverge: unchecked
- Reform each timestep: unchecked

This completes the tutorial. The figure below shows the results of the model (Figure 2.29).

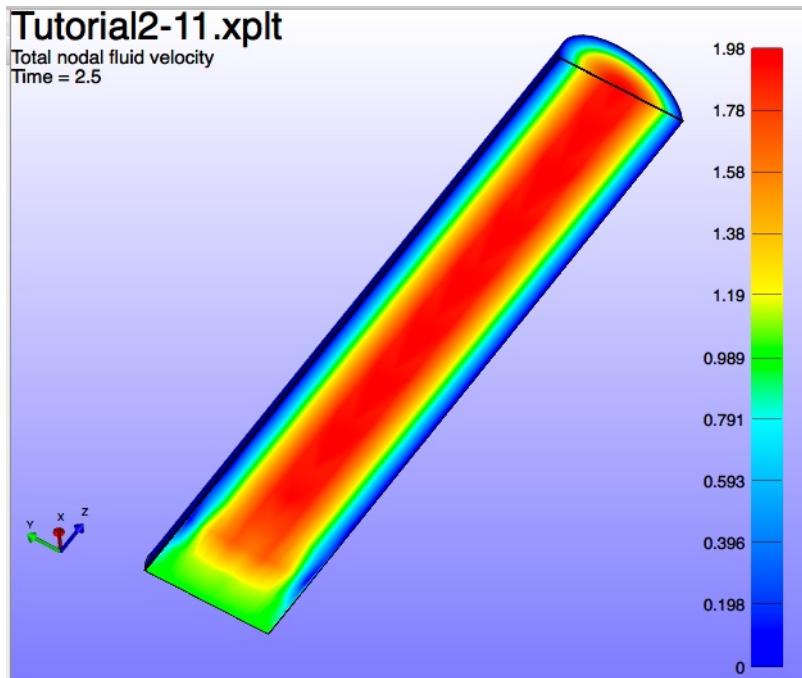


Figure 2.29: The result for Tutorial 11 shown in PostView. There is a plane cut on the x-plane to show the solution on the interior of the mesh.

2.12 Tutorial 12: Setting up a Fluid-Structure Interaction Analysis

This tutorial will illustrate how to set up a fluid-structure interaction (FSI) problem. Similar to the fluids problem, a simple pipe flow problem will be created, where now there is a deformable elastic wall surrounding the fluid domain. We select boundary conditions consistent with the assumption that our finite element domain models only a truncated section of a longer pipe. Therefore, a fully-developed parabolic normal fluid flow is prescribed at the inlet and pressure resistance with a pressure offset is prescribed at the outlet. Remember to Edit Project Settings and pick the Mechanics, Fluid, and Fluid-FSI modules when PreView is opened or when starting a new project.

2.12.1 Step 1: Creating the Geometry and Mesh

The geometry of this problem is a cylinder. Activate the *Create* panel and create a cylinder located at the origin (0,0,0) with parameters *Radius* = 1, *Height*=10. Then activate the *Mesh* panel. Enter the following settings:

- Slices = 8
- Segments = 12
- Stacks = 40
- R-bias = 0.75
- Mesh Type = Wedge center

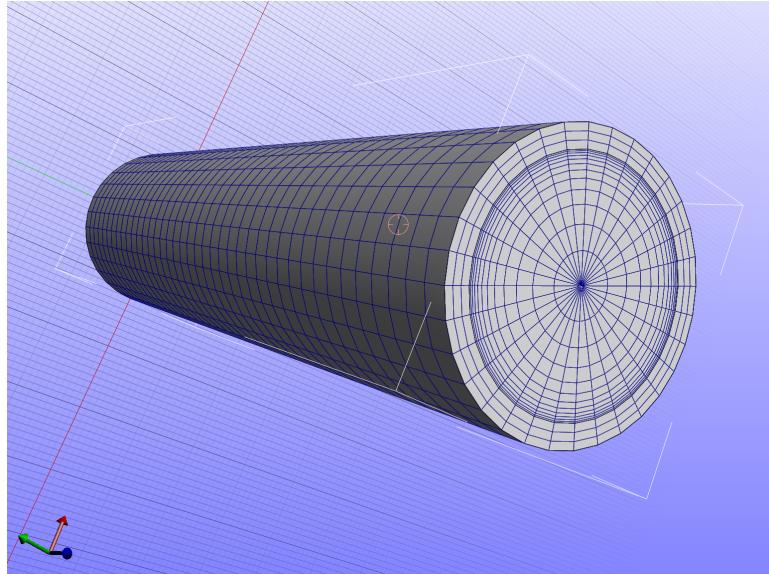


Figure 2.30: The model geometry and mesh for Tutorial 12.

Leave other parameters at their default values and click the *Apply* button. Now we create the cylinder wall by extruding the cylindrical surface. Click the drop-down menu labeled *Convert* and select *Editable mesh* in order to perform the extrusion. Making sure the cylinder object is selected, click the *Select faces* button on the graphical toolbar under the graphics viewing window. Select all of the faces on the lateral cylindrical surface (don't include the end faces), taking advantage of the *Select connected* option. Under the *Edit Mesh* menu, click the *Extrude Faces* button. Enter the following settings:

- Distance = 0.2
- Segments = 3
- Use local normal = checked

Then click the *Apply* button. The cylinder wall should have been created. The geometry should look as follows in Figure 2.30.

2.12.2 Step 2: Setting up the materials

In this model we use a Newtonian fluid. Since the fluid domain may expand as the fluid pressurizes, we use a fluid-FSI material which includes a solid component to account for the mesh deformation of the fluid domain. The solid component should have zero density and a very low stiffness; in principle you can choose any of the available elastic materials in FEBio, however the material that works best for FSI analyses is the neo-Hookean material model with its Poisson ratio set to zero. To add a material to the model, open the *Material Browser* by selecting the *Physics/Add Material* menu. Alternatively you can also open the *Material Browser* by right-clicking on the *Materials* item in the Model Viewer and selecting *Add Material* from the popup menu. There is also a shortcut for this on the toolbar (icon with four colored circles). First, from the Category option select *fluid-FSI* and name this material 'Fluid domain'. Then, select the *fluid-FSI* material, the *fluid* model, and then the *Newtonian fluid* viscous model. For the solid material select *neo-Hookean* and click

OK. The newly added material should now be highlighted in the Model Viewer and the material properties are now displayed in the properties panel, located below the Model Viewer. With the material selected in the Model Viewer you will notice that it has a selection box that lists the parts to which this material is assigned. Making sure to only select the fluid domain (the original cylinder) in the Graphics View using the *Select Parts* option, click on the plus-sign in the material's selection box, to assign this material to the part. The color of your object will now change to indicate that it is now associated with the fluid-FSI material. Then, in the model tree, expand the fluid-FSI material and click on *fluid* and set the following material properties for the fluid:

- density = 1
- bulk modulus = 1e+09

Now expand the fluid material and click on *viscous*. Set the following material properties:

- shear viscosity = 0.1
- bulk viscosity = 0

Click on *solid* and set the material properties:

- density = 0
- Young's modulus = 1e-09
- Poisson's ratio = 0

Note that FEBio will automatically reset the density of the solid in a fluid-FSI material to zero, even if you specify a non-zero value, since this is an underlying assumption of the FSI formulation. Now we need to set the material properties of the cylindrical elastic wall. Going back to *Add Material*, under the *uncoupled elastic* category, name the material 'Elastic wall' and choose the *Mooney-Rivlin* material and click *OK*. With the material selected in the Model Viewer, select the solid wall part and add it to the material selection. Set the following material properties for the solid:

- density = 1
- c1 = 1e+05
- c2 = 0
- bulk modulus = 1e+07

2.12.3 Step 3: Setting up the boundary conditions

To impose a no-slip condition at the interface between the viscous fluid and the elastic wall, select that interface and apply a zero fluid velocity boundary condition in the *x*, *y*, and *z* degrees of freedom. In actuality we are fixing the relative velocity between the fluid and wall, since the elastic wall may also deform. This can be done by selecting *Physics* from the menu and then selecting *Add Boundary Condition*. Alternatively, on the model tree you can right-click *Boundary Condition* and select *Add Boundary Condition*. Or another way to bring up the boundary condition menu is to select hotkey Ctrl-B (Cmd-B on Mac). Now, in the add BC menu, select *zero fluid velocity* and press *OK*. The boundary condition should appear in the model viewer. Under *Properties* click the check boxes for *x*, *y*, and *z* fluid velocity since we want to fix them. Now, making sure that the

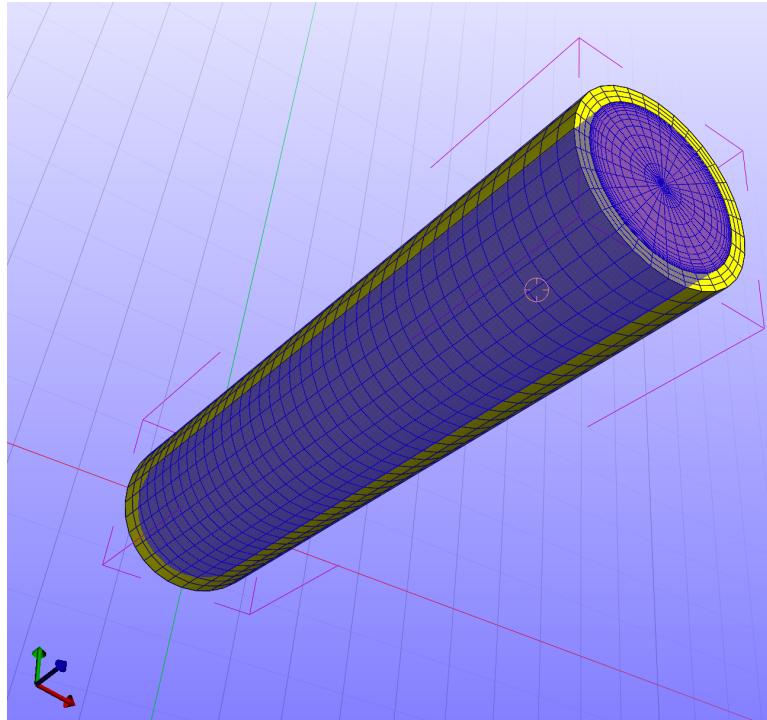


Figure 2.31: The interface boundary for the model from Tutorial 12.

Select Surfaces button is clicked, select the interface between the fluid and elastic solid domains, which will likely require you to click twice since the first click will select the outer solid wall surface; then click the plus-sign in the BC selection box, so that this BC can be assigned to the selected surface (Figure 2.31).

Next, we need to suitably fix the displacement of the solid domains. Go back to *Add Boundary Condition* and select *Fixed Displacement*. Make sure the surface previously highlighted is deselected by clicking anywhere outside the model in the graphics viewer. In the model viewer, under *Properties*, check the box for *z-displacement* as we are assigning a symmetry boundary condition to the ends of the cylinder. Assign this boundary condition to the end faces of the solid elastic wall and the fluid-FSI domains, for a total of four surfaces.

Since we will be prescribing a parabolic fluid velocity profile at the inlet face of the fluid domain, FEBio can produce more stable numerical convergence if we prescribe or fix the fluid dilatation on the circular edge of that inlet face (the face whose outward normal is along the negative z -direction). Make sure that *Select Curves* is selected and assign a *zero fluid dilatation* BC to that circular edge. This boundary condition is needed since the dilatation on the inlet curve would otherwise be indeterminate.

Now we have to prescribe the surface loads. This time, select *Physics* from the menu and then select *Add Surface Load*. Alternatively, on the model tree you can right click *Loads* and select *Add Surface Load*. Or another way to bring up the boundary condition menu is to select hotkey Ctrl-L (Cmd-L on Mac). Now, from the *Add Surface Load* menu, click on the option *fluid normal velocity* and press *OK*. The surface load should show up in the model viewer. Under *Properties*, we want to set the *velocity=-1* as the fluid is to flow into the cylinder (opposite to the outer surface normal). The *parabolic* option should be checked. Then keeping all the other parameters at their default value, assign this boundary load to the fluid inlet surface.

Next, we want to prescribe outlet boundary conditions. Going back to *Add Surface Load*, now select *Fluid Flow Resistance*. Under *Properties*, let us set *resistance=1000* and *pressure_offset=100*. Assign this boundary load to the fluid outlet surface, which is the cylinder end face not on the origin. Note that for both the inlet prescribed velocity and outlet resistance and pressure, a load curve can be assigned that controls the magnitudes of these boundary conditions over time if so desired. The *Curve Editor* can be accessed from either selecting *Tools* and then *Curve Editor* from the menu or by clicking the *Curve Editor* button from the toolbar. However, for this tutorial, the default load curves will be used.

Lastly, we need to make sure the fluid pressure is felt by the elastic wall. Going again to *Add Surface Load*, now select *FSI Interface Traction*. Assign this boundary load on the interface side wall between the fluid and solid domains. Again, like before the surface may have to be clicked twice such that this inner surface is correctly selected (Figure 2.31). No parameters are needed for this boundary load.

2.12.4 Step 4: Setting up the analysis step

Finally, an analysis step needs to be defined. Select the *Physics/Add Analysis Step* menu. Select *Fluid-FSI Mechanics*. Click *OK* to add the step to the model. Then change the following parameters under the Model Viewer while keeping all other parameters the same:

- Time steps = 30
- Optimal iterations = 50
- Matrix symmetry = non-symmetric
- Residual tolerance = 0.001
- Max Reformations = 5
- Max updates = 50
- Reform on diverge: unchecked
- Reform each timestep: unchecked

Save the file, export the model and run the analysis. You will notice that the elastic wall inflates a little bit as a result of the fluid pressurization.

This completes the tutorial. Figure 2.32 shows the results of the model.

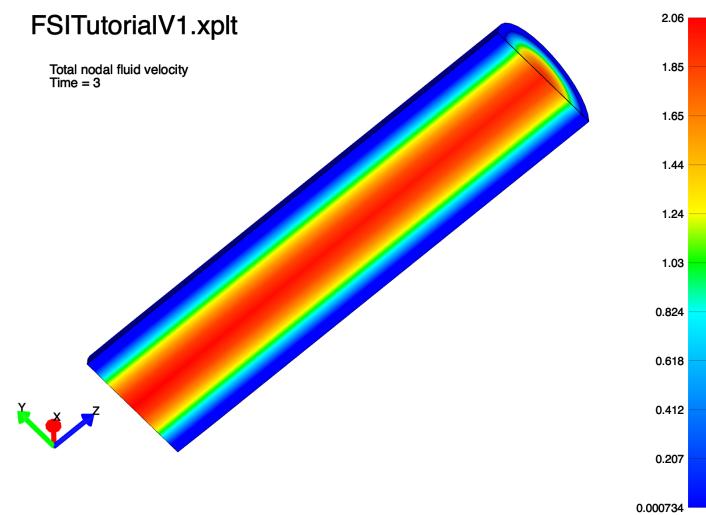


Figure 2.32: The result for Tutorial 12 shown in PostView. There is a plane cut on the x-plane to show the solution on the interior of the mesh.

Chapter 3

The PreView Environment

This chapter provides an in-depth description of PreView’s graphical user interface and how to interact with it.

3.1 The Graphical User Interface

3.1.1 Overview

PreView has a powerful graphical user interface (GUI) that offers an intuitive approach to setting up a finite element problem. It has several components to it, and in order to make optimal use of PreView it is important that the user is familiar with PreView’s GUI. Tutorial 1 guides the user through the most important skills needed to manipulate the GUI. In this chapter we provide a more in-depth discussion of it.

Figure 3.1 shows the PreView GUI and many of its most important components. The *Main Menu* bar gives access to most of PreView’s features such as file I/O, editing selections, setting up the physics, customizing the view and much more. The *Main Toolbar* provides an alternative way to invoke some of the most commonly used menu commands. It also has some buttons that affect the way the user interacts with the model. The *Model Viewer* shows a hierarchical overview of the model and its components. The *Graphics View* displays a 3D view of the model. The *Status bar* at the bottom of the screen displays some information regarding PreView’s status. The *Graphics Toolbar* shows additional tools for interacting with objects and selections in the Graphics View. The *Build panel* has several child panels that are used to select the current active working context. The *Create panel* allows the user to create geometry, the *Edit panel* lists several options to select and edit the geometry. The *Mesh panel* collects all parameters that affect mesh generation. Finally the *Tools panel* offers some alternative editing tools.

3.1.2 Navigating the GUI

Navigating the GUI is done using the mouse. By holding down one of the three mouse buttons and dragging the mouse, the viewing position can be changed. The view can be rotated, panned (i.e. translated), or zoomed. The following table gives an overview of the different methods to change the view.

In the following sections, the different components of the menu bar will be explored.

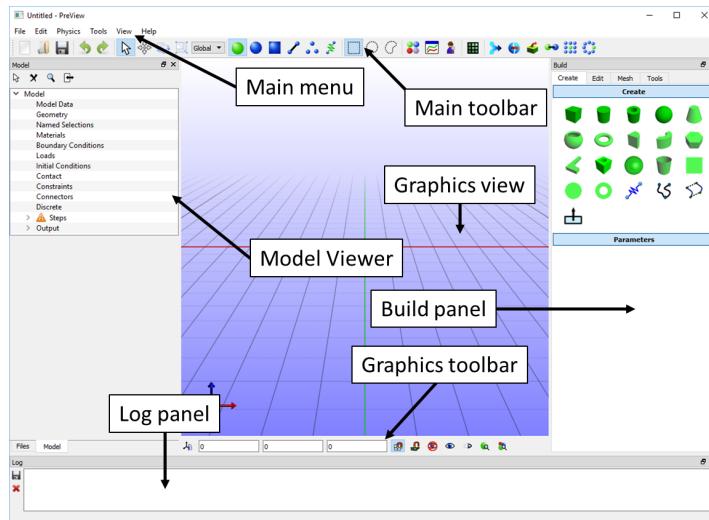


Figure 3.1: PreView's Graphical User Interface.

Action	Standard Method	Alternative Method
Rotate	left mouse button	
Rotate in-plane	left mouse button + Alt key	
Panning	middle mouse button	right mouse button + Alt key
Zooming	right mouse button	

Table 3.1: Overview of methods to change the view.

3.2 The Menu Bar

The *Menu Bar* found at the top of the window gives access to the following menus:

- *File*: Open files, save files, export or import files, etc.
- *Edit*: Undo/Redo feature, options to edit the selection.
- *Physics*: Define boundary conditions, contact interfaces, materials , etc.
- *Tools*: provides access to a set of useful tools, including the Options dialog.
- *View*: modify view settings and access additional windows.
- *Help*: Access to PreView's help menu and About box.

A more detailed explanation of the available menu items follows.

3.2.1 The File Menu

The File menu offers the following menu items:

- *New*: Start a new PreView model. This will start the New Project Template dialog where you can select a project template.
- *Open*: Open a PreView model from file.

- *Save*: Save the current PreView model to file.
- *Save as*: Save the current PreView model under a different name.
- *Recent files*: Shows a list of recently accessed PreView files.
- *Import FE Model*: Import a finite element model from a particular file format. This usually starts a new project as well.
- *Export FE Model*: Export the current project to a FE file format.
- *Recent FE Model files*: Shows a list of recently accessed FE model files. Selecting a file here will usually start a new project as well.
- *Import Geometry*: Import a surface or volume mesh into the current project.
- *Export Geometry*: Export the mesh of the selected geometry to a surface or volume mesh file format.
- *Recent Geometry Files*: Shows a list of recently accessed mesh files.
- *Batch convert*: Convert a list of files from one format to another. (Currently only supports converting between an older to a newer FEBio file format.)
- *Exit*: Exits the application.

3.2.2 The Edit Menu

The Edit menu offers the following menu items:

- *Undo*: undo the last operation
- *Redo*: redo the last undone operation
- *Invert selection*: Invert the current selection
- *Clear selection*: Clear the current selection
- *Delete selection*: Delete the current selection
- *Name selection*: name the current selection
- *Hide selection*: hide the current selection
- *Unhide all*: Shows all the hidden objects.
- *Toggle Visibility*: Hides visible objects and parts and shows hidden ones.
- *Transform*: Modifies the current selections position, rotation, and scale.
- *Collapse Transform*: Applies the selected object's transformation to its mesh and resets its transform.
- *Clone object*: create a copy of the currently selected object.
- *Merge objects*: Merges all the selected objects into a single editable mesh.

- *Purge*: removes all physics from the model (materials, BC's, loads, etc.)
- *Edit Project Setting*: Change the settings of the active project. This allows users to activate or deactivate the modules.

3.2.3 The Physics Menu

The Physics menu offers the following menu items:

- *Add Boundary Condition*: Apply a boundary condition to the selection.
- *Add Nodal Load*: Apply a load on a selected node.
- *Add Surface Load*: Apply a boundary load to a surface.
- *Add Body Load*: Apply a body load.
- *Add Initial condition*: Define an initial condition for the selection.
- *Add Contact*: Set up different contact conditions between the objects in the model.
- *Add Rigid Constraint*: Define a constraint to a rigid body.
- *Add Rigid Connector*: Define a rigid connector between two rigid bodies.
- *Add Material*: create a new material and manage material libraries.
- *Add Analysis Step*: define a new analysis step.
- *Solute Table*: Define all solutes for this analysis.
- *Solid-bound Molecule Table*: Define all the solid-bound molecules of this analysis.
- *Chemical Reaction Table*: Define all the chemical reactions of this analysis.

3.2.4 The Tools Menu

The *Tools* menu offers the following menu items:

- *Curve Editor*: Show the Curve Editor tool
- *Mesh Inspector*: Activate the Mesh Inspector tool.
- *Elasticity Convertor*: convert between parameter for defining elastic materials.
- *Run FEBio*: call the external FEBio solver.
- *Options*: Open a dialog box that allows you to edit PreView's settings.

3.2.5 The View Menu

The *View* menu offers the following menu items:

- *Undo view change*: undo the last view change
- *Redo view change*: redo the last undone view change
- *Zoom to Selection*: Zooms in on the current selection.
- *Orthographics Projection*: Toggles between orthographic or perspective projection mode.
- *Show Normals*: Toggles surface normal on or off of the active object.
- *Show Grid*: toggle the grid in the GV on or off.
- *Show mesh lines*: toggle the mesh lines in the GV on or off.
- *Show Edge lines*: toggle the rendering of the crease edges on or off.
- *Backface culling*: toggle backface culling on or off.
- *Front*: show the front plane in orthographic projection mode.
- *Back*: show the back plane in orthographic projection mode.
- *Right*: show the right plane in orthographic projection mode.
- *Left*: show the left plane in orthographic projection mode.
- *Top*: show the top plane in orthographic projection mode.
- *Bottom*: show the bottom plane in orthographic projection mode.
- *Files*: Toggles visibility of the Files panel.
- *Models*: Toggles visibility of the Models panel.
- *Build*: Toggles visibility of the Builds panel.
- *Log*: Toggles visibility of the Log panel.

3.2.6 The Help Menu

The Help menu offers the following menu items.

- *Online Help*: Opens the PreView Help window.
- *About*: Displays the PreView About Box.

3.3 The Main Tool Bar

Some of the menu items can also be accessed through the Main Tool bar. The Main Tool bar offers the following options.

-  Start a new PreView project.
-  Open a saved PreView project.
-  Save the current PreView project.
-  Undo the last operation
-  Redo the last operation
-  Enter selection mode
-  Enter select-and-move mode
-  Enter select-and-rotate mode
-  Enter select-and-scale mode
-  Global Set the transformation coordinate system
-  Select Objects
-  Select Parts
-  Select Faces
-  Select Edges
-  Select Nodes
-  Select Discrete Objects
-  Rectangle Selection Mode
-  Circle Selection Mode
-  Freehand Selection Mode
-  Display the Material Browser



Display the Curve Editor



Activate the Mesh Inspector tool



Toggle Mesh Lines



Clone Object



Clone Grid Object



Clone Revolve Object

3.4 The Graphics View

The Graphics View shows a 3D view of the current model. The user can adjust the view in several ways. The fastest way of maneuvering through the view is by using the mouse. The different mouse buttons invoke different commands depending on the current *mode* of the Graphics View. There are four buttons on the Toolbar that control this mode, namely the *select*, *move*, *rotate* and *scale* buttons. The current mode is indicated by the button that is highlighted.

Regardless of the Graphics view mode, the view can be rotated by dragging the mouse button outside any geometry while holding down the left mouse button. A similar action but with the right mouse button down allows the user to zoom the view. The user can pan the current view by holding down the middle mouse button. Objects and items can be selected in all four Graphics view modes. To select an object or item (e.g. part, surface, edge), bring the mouse cursor over the object or item, and click on it with the left mouse button. To add objects or items to the current selection, hold down the shift button while clicking on the object or item. To add several objects or items, drag the mouse while holding down the shift button and left mouse button. To deselect objects or items, the same operations can be performed, only this time, hold down the ctrl button instead of the shift button. To deselect all objects or items at once, click in an empty area of the Graphics View. There are also several menu items that allow you to modify the current selection. See section 3.2.1 for more information.

By default, PreView renders the model using *perspective projection* in the Graphics View. By right-clicking with the mouse button a popup menu shows up under the cursor that allows the user to toggle between perspective and orthographic projection. In addition, this menu provides some buttons that allows the user to quickly orient the geometry along one of the three coordinate axes.

When the Graphics View is in one of three transform modes (*move*, *rotate* or *scale*) a triad will appear in the center of the selection. When moving the mouse over the triad, one of its arms may highlight, indicating that this arm can be selected. To apply a transformation to the current selection, bring the mouse cursor over one of the triad arms so that it highlights. Next, drag the mouse button while holding down the left mouse button. Releasing the left mouse button will let go of the arm and finish the transformation.

3.5 The Graphics Toolbar

The *Graphics Toolbar* can be found at the bottom of the Graphics View and provides the user with some information regarding the Graphics View and some additional tools that affect the current selection.



This toggles the *pivot lock* mode. The pivot is the point that serves as the origin for transformations. Usually it is calculated automatically based on the current selection, but by clicking this button, the user can lock the pivot and edit its coordinates manually in the next three panels.



When the *snap-to-grid* button is checked, a selection can only be translated in increments of the current grid spacing.



This option allows users to position the 3D cursor at nodes of the geometry.



This toggles all visible objects and parts. (Or elements when in mesh selection model.)



Zoom to the current selection.



Zoom to the extents of the current model.

3.6 The Model Viewer

The *Model Viewer* shows a hierarchical overview of all the components of the model and their interdependencies. The model data is organized in separate categories, depending on the purpose of the data.

- *Model Data*: Displays the global model parameters.
- *Geometry*: Contains a list of all the objects in the model.
- *Named selections*: Contains all the user-defined selections. These named selections can be used to easily refer to a collection of parts, surfaces, edges, or nodes and are useful for defining boundary conditions and contact interfaces.
- *Materials*: Lists all the materials defined for the model.
- *Boundary Conditions*: Lists all the boundary conditions defined for the model and for all analysis steps.
- *Loads*: Lists all the loads that are defined for the model and for all analysis steps.
- *Initial Conditions*: Lists all the initial conditions for the model and for all analysis steps.
- *Contact*: Lists all the contact interfaces that are defined for the model and for all analysis steps.
- *Constraints*: Lists all the rigid constraints defined for the model and for all analysis steps.
- *Connectors*: Lists all the rigid connectors defined for the model and for all analysis steps.
- *Steps*: Lists all the steps that are defined for this model. Each step will be listed as a sub-item of this item. Each sub-item has additional items that list all the components that will be active only during that particular step.

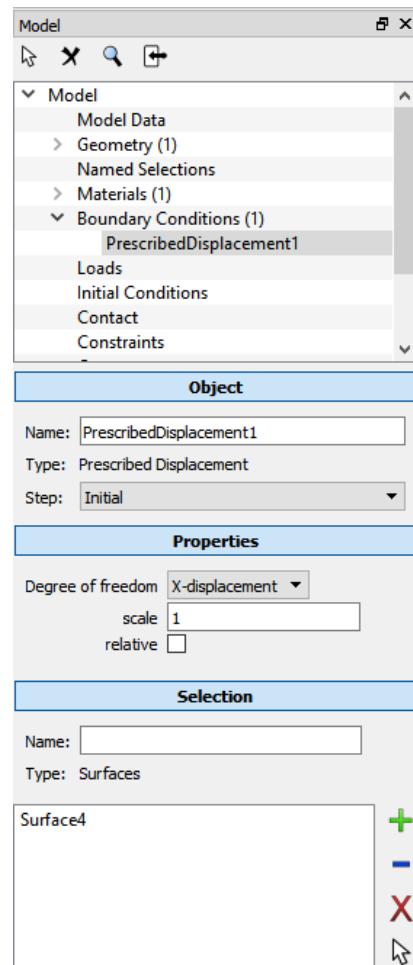


Figure 3.2: The Model Viewer shows a hierarchical overview of the model components.

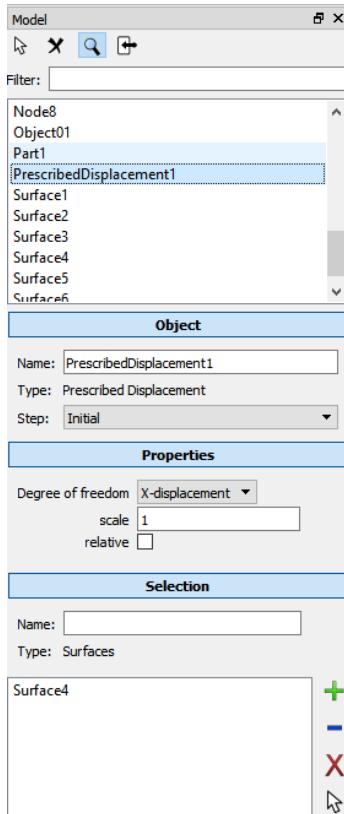


Figure 3.3: The Model Viewer with the Search Panel active.

- *Output:* Can be used to define the field variables that need to be output to the FEBio plot file.

Many of the PreView features can also be accessed by right-clicking on an item in the Model Editor. A popup menu will appear that lists the available options for that particular item. For example, you can add a new material by right-clicking on the Materials item and selecting *Add Material...* from the popup menu.

At the top of the Model Editor additional buttons are located that allow access to the following features.

- For model components that have associated geometry items, this button selects these items in the Graphics View.
- Delete the currently selected items in the Model Viewer.
- Activate the Search panel, which provides an alternative method for selecting items in the Model Viewer.
- Clicking this button will synchronize the selection between the Graphics View and the Model Viewer. This only works for geometry items.

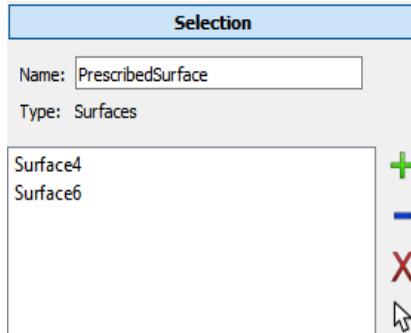


Figure 3.4: The selection pane allows users to edit the geometry selections assigned to certain physics components.

3.6.1 The Search Panel

The Search panel is an alternative method for inspecting model components. When active, users can enter a search filter at the top. All items with a name that contains the filter will be displayed in the box below the filter. Users can select items here and the properties will be displayed below. Users can also right-click on items and select an option from the popup menu. Double-clicking on an item will toggle back to the tree view with that item selected.

3.7 The Model Viewer Panels

Below the Model Viewer you will see several panels that provide information about the item that is active in the Model Viewer's tree view. The particular panels that are shown depend on the selected item.

- *Object*: Shows the name and type of the active item. For physics components this also shows the step in which that component is active.
- *Properties*: This panel displays the editable parameters of the item that is selected in the Model Viewer.
- *Selection*: This panel is shown for all components that can have a selection assigned to it, such as materials, named selections, boundary conditions, and loads. Contact interfaces will have two selection panels, one for the master and one for the slave surface. Detailed instructions on how to edit the selection can be found in section 3.7.1.

3.7.1 Editing Selections

Some components of the model, such as materials, named selections, and boundary conditions, need to be applied to a selection of the geometry. These components will have a selection box in the Model Viewer to which items of the model can be assigned to. For instance, for materials, it will show the parts of the model to which the material is assigned.

The list that is displayed in the selection box can be edited using the buttons located on the right side of the selection box. Note that you can select the items in the selection box.

 This button adds the current selection in the Graphics View to the list. Note that if the list was not empty, then the selection will be added if it has the same type as the items already in the list. For instance, if you applied a boundary condition to a surface, you can only add (or remove) surfaces from its list.

 This button will remove the current selection in the Graphics View from the list.

 This button removes the items selected in the list from the box.

 This button selects the items that are selected in the box also selected in the Graphics View.

3.8 The Build Panel

The Build panel allows users access to the geometry and mesh editing tools in PreView. It consists of several child panels, each offering a different set of model editing tools.

3.8.1 The Create Panel

The Create panel is used to create geometry. PreView offers a basic set of mesh generation features; you can create *primitive* geometries, such as boxes, cylinders, spheres and so on. By activating the *Create* panel a list of buttons is displayed. By clicking on one of the buttons, the creation parameters appear for a particular primitive. Once you have entered the parameters you can click the *Create* button to add the primitive to your model. See section 4.4 for more details on how to create geometry.

3.8.2 The Edit Panel

The *Edit* panel allows you to modify the object creation parameters. By modifying these parameters, the user can change the geometry at any time. See section 4.3 for more details on how to edit geometry.

3.8.3 The Mesh Panel

The *Mesh* panel allows the user to set the meshing parameters of the geometry. By modifying these parameters, the user can control the number and distribution of elements in the mesh. The available parameters depend on the selected geometry. For *Editable Meshes* this panel will show a set of tools that allow the mesh to be modified.

3.8.4 The Tools Panel

This panel defines a few specialized tools that can be useful for very specific tasks.

3.9 The Curve Editor

The *Curve Editor* is accessed either from the *Tools/Curve Editor* menu or by pressing the corresponding button on the toolbar. It can also be accessed using the *F4* shortcut. The *Curve Editor*

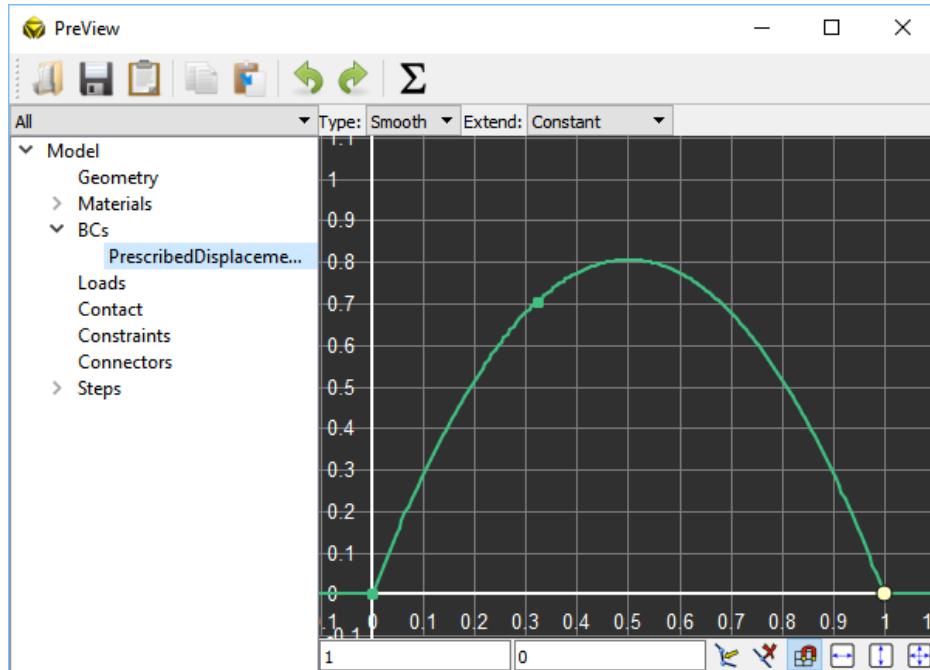


Figure 3.5: The curve editor shows all the load curves used in the model.

gives an overview of all the time-dependent parameters in the model. PreView allows the user to define the time dependency explicitly through the use of so-called *load curves*. A load curve is simply an interpolated function of (time, value) pairs. The value will have a meaning that depends on the parameter associated with the curve. For example, if the load curve describes a nodal displacement, the value of the curve is the actual displacement and has the length units used in the design of the geometry. If the load curve describes a material parameter, then the value is the time evolution of that parameter.

It is important to note that many features that allow you to define a load curve also allow you to define a scale factor. In that case, the actual value of the parameter is the scale factor multiplied with the load curve value. For example, when prescribing the displacement, the scale factor is defined in the properties dialog when creating the prescribed constraint. The scale factor is not taken into account in the Curve Editor.

At the top of the curve editor's overview panel, the selection filter can be found. This filter allows you to show the load curves of only a specific type of model component, e.g. boundary conditions, material parameters, etc. This filter can be helpful in finding a specific parameter.

Most load curves have a default linear shape that ramps up the value from zero to one. However, all curves can be modified using the tools available at the bottom of the Curve Editor. The curve's data points are represented as dots on the view. These data points can be selected by clicking on them with the left mouse button. They can also be moved by click+dragging them. The current (time, value) pair of the selected point is displayed on the toolbar at the bottom of the view. This toolbar offers the following features.

When this button is toggled, each click on the curve view will add a new node. Note that you don't click this button to add nodes. Nodes can also be added by shift+click with the left mouse button on the curve view.

This button will delete the node that is currently selected in the curve view.



The snap-to-grid option will allow you to move a node on the intersections of the grid lines.



Zooms the curve view out so that all nodes are visible within the bounds of the curve view. The toolbar at the top of the Curve Editor provides additional tools for modifying the active curve.



This will load data from a file. The file must be a simple text file with one line of data for each point. On each line, specify the time-load value pair delimited by a space. You can enter as many lines as you want.



Save the active load curve to a text file.



Copy the curve data to the clipboard. This allows the curve data to be pasted in another application that supports clipboard operations.



Store the load curve data so it can be pasted to another curve.



Paste the curve data that was copied to the active load curve.



Undo the last change to the active load curve.



Redo the last change that was undone.



Open the equation editor where load curve data can be generated via a mathematical equation (see below).

The current view can be zoomed in or out, either by using the zoom buttons at the bottom of the view, or using the mouse wheel. When you scroll the mouse wheel while hovering over one of the axes, the graph will only zoom in that axis' direction. Depressing the left mouse button while moving the mouse, pans the view.

At the top of the curve view, you can see two drop-down lists. The first list allows the user to set the curve type which defines the interpolation mode for the currently displayed curve. The choices are as follows:

- **linear**: use a linear interpolation between the curve points
- **step**: use a constant interpolation between the curve points. The value of the curve is defined by the value of the point closest and to the right of a particular ordinate.
- **smooth**: A cubic spline is fitted through the data points.

The second list displays the *extend mode* options. The extend mode defines the value of the curve outside its defined range. The choices are:

- **Constant**: the value is clamped to the range of the curve as defined by the first and last point.
- **Extrapolate**: the value is extrapolated from the end-points of the curve.
- **Repeat**: the curve is repeated on either end of the curve's domain

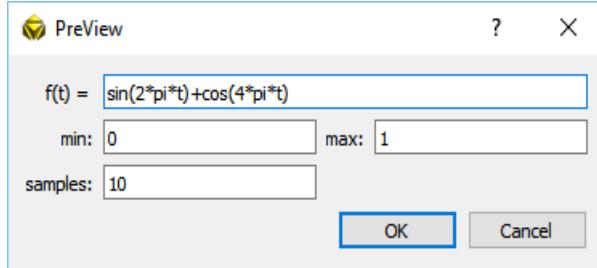


Figure 3.6: The equation editor allows user to easily generate load curve data via a mathematical expression

- **Repeat offset:** same as repeat, except that the curve is offset by the end-point values.

If the previous tools are not sufficient to describe the evolution of the load curve in detail, the *Equation Editor* can be used (Figure 3.6). This tool is accessed from the toolbar and allows the user to enter a mathematical function of time. Use the symbol *t* to reference time. This function will be evaluated and discretized to generate a set of points that interpolate the function approximately.

3.10 The Mesh Inspector

The Mesh Inspector window displays some statistics of the mesh of the currently active object (Figure 3.7). It shows, for instance, what elements the mesh is composed of and the user can inspect element metrics such as element volume, jacobian, etc.

The mesh info pane at the top lists the total number of nodes, faces, and elements in the current mesh. It also shows a list of all the different element types and their count.

The variable pane allows the user to select the quality metric that is displayed in the bar chart. To limit the plot only to a certain type of element, select the corresponding element in the element list of the mesh pane.

The statistics pane shows the min, max and average values of the selected variable. The selection pane can be used to select elements in a certain range of the currently plotted variable. This can be useful to identify and quickly select elements of interest.

3.11 PreView Options

The PreView Options dialog can be opened from the *Tools/Options* menu or by pressing the F6 shortcut button. Different categories can be selected by clicking the tab buttons. By selecting a category a list of options is shown on the right hand side of the dialog box. The categories are:

- *Background*: change the background settings such as color, gradient, ...
- *Display*: Shows options that affect how objects and meshes are rendered in the Graphics View.
- *Physics*: Allows the user to select options that draw physics related items in the Graphics View.
- *UI*: Shows options that affect how users interact with some UI components.

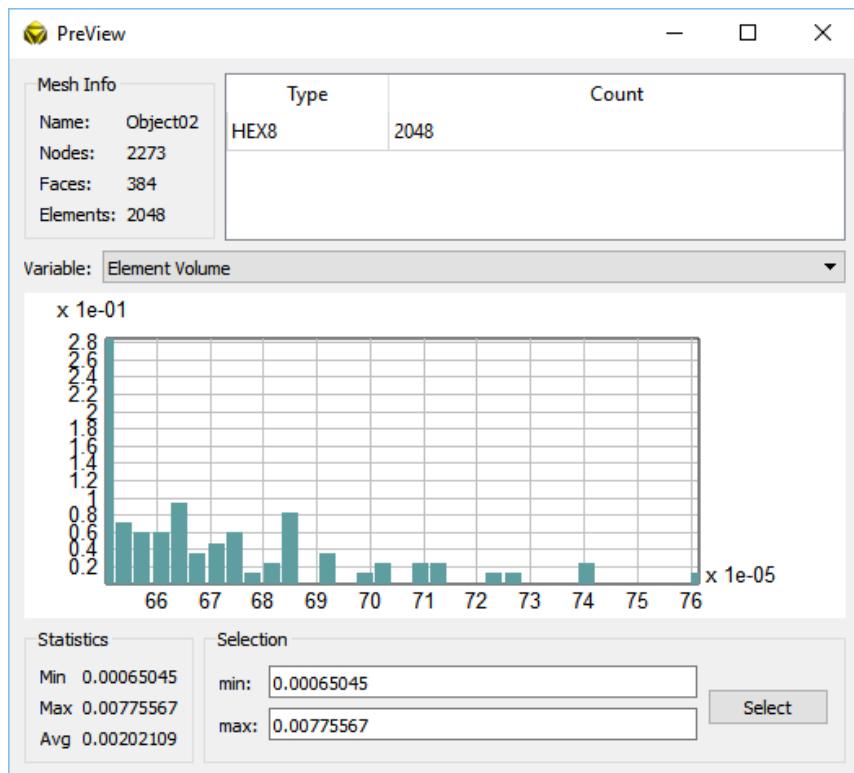


Figure 3.7: The Mesh Inspector tool allows the user to inspect certain quality measure of the mesh.

Chapter 4

Creating, Loading, and Saving Projects

4.1 Starting a new project

A new project can be started from the *File\New...* menu or by clicking the corresponding toolbar button. This will open the New Project dialog box. This dialog presents the project templates that are available. A project template is a collection of modules that will be activate in a project. A module corresponds to certain physics modeling features or UI components. Project templates and modules greatly simplify the process of creating a FE model by presenting only relevant features to the user.

Choosing Cancel on this dialog will activate all modules and expose all the physics features and modeling capabilities. You can also change the active modules after starting a new project from the menu *Edit\Project Settings*.

4.2 Loading a project

PreView project files (extension *.prv*) can be loaded from the *File\Open* menu. A standard file open dialog box appears where a file can be selected and opened.

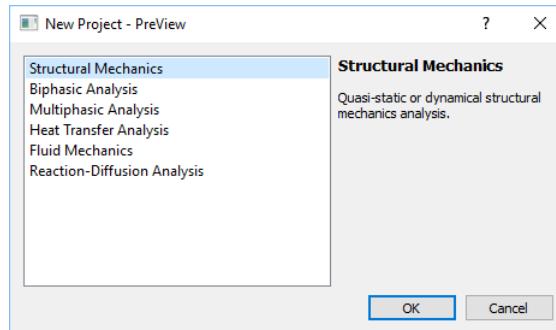


Figure 4.1: The New Project dialog allows users to choose a project template.

4.3 Saving a project

A project can be saved from the *File\Save* or *File\Save as* menus. The former can be used to save the file to the same location as it was loaded. For new projects that have not been saved yet, this will execute the *File\Save as* command on the first attempt to save the project. The *File\Save as* menu allows the user to select a file name for storing the project.

4.4 PreView Special File Formats

Besides the PreView project file format (.prv), PreView offers two additional file formats that allow users to save and load parts of a PreView project. These special files formats are discussed next.

4.4.1 PreView Object File

The PreView Object file format (extension .pvo) allows users to store geometry and meshes created in PreView to an external file. This is useful for storing objects that need to be reused often.

To save a PreView object to a pvo file, first select the object (or objects) in the Graphics Viewer. Then, select the *File\Export Geometry* menu. If the Save dialog box, select the “PreView Object” filter and click OK. This will store all the selected objects to a pvo file.

PVO files can be imported into an existing PreView project. First, select the *File\Import Geometry* menu. In the Open dialog box select the “PreView Object” file filter and then select your files. Next, click OK and PreView will load all the objects in PVO file into the current project.

4.4.2 PreView Material File

Users can export material definitions to an external file, called the PreView Material file. To do so, right-click on the *Materials* item in the Model Viewer and select the *Export Materials* item from the popup menu. A File Open dialog box appears where the PVM file can be selected. Next, click OK to export all the materials.

If you only want to export some of the materials, select the materials in the Model Viewer (You can select multiple materials by shift+clicking on each material), then right-click one of the selected materials and then select *Export Material(s)* from the popup menu.

To import materials from a PVM file, right-click on the *Materials* item in the Model Viewer and select *Import Materials* from the popup menu. A File Open dialog appears where the PVM file can be selected. Press OK to import all the materials in PVM file.

Chapter 5

Creating and Editing Geometry

5.1 Creating Geometry

Although PreView is not designed to be a mesh generator, it has a few features to create simple geometries and meshes. Most of these options are available from the *Create* panel. At the top it shows the various creation options.

- *Primitives*: Create simple geometric shapes.
- *Discrete Objects*: Create springs
- *CAD Objects*: Create geometry via CAD-like generation methods.

The most commonly used creation option is *primitives*. When selected a list of available primitives is shown. When selecting a primitive, its creation parameters are shown. After entering the desired values, click the *Create* button to create the primitive and add it to the model. The new object will be automatically selected in the Graphics View and shown in the Model Viewer.

5.2 Importing Geometry

Instead of creating geometry directly in PreView it is more common to create your geometry in dedicated software and import the mesh into PreView. PreView can import several different mesh file formats. Geometry can be imported using the *File/Import Geometry* menu. Appendix A lists an overview of all the supported file formats including the supported features of the particular formats.

5.3 Editing Geometry

The *Edit panel*, one of the panels on the Build panel, gives access to the object's creation parameters and editing options. For primitives this shows the same parameters as when the object was created. This allows you to change the object's dimensions at any time. If the object is not a primitive, no options will be shown here.

The other feature to note at the top of both the Edit and Mesh panels is the Object rollout. This shows the name of the active object as well as its type. An object can be converted to a different type via the *Convert* button. It offers two options:

- *Editable surface*: An object of this type will have a surface mesh that can be modified. A separate volume mesh can be generated for this object.
- *Editable mesh*: An object of this type will only contain a volume mesh and this mesh can be modified if necessary.

Editable surfaces and editable meshes will be explained in detail in section 5.4.

5.4 Creating and Editing a Mesh

The *Mesh panel* gives access to PreView's meshing capabilities. As noted before, PreView was not designed to be a powerful mesh generator, but regardless, has some simple mesh creating and editing features.

5.4.1 Meshing Primitives

When the object is a primitive, that is, created with one of the geometry creation tools, the command window will list the mesh creation parameters for that particular object. Most primitives will be meshed with a so-called *butterfly mesh*. These primitives have a simple rectangular box as center. The rest of the mesh is a projection from this box to the respective geometry. For instance, for the sphere the surface of the inner box is projected onto a sphere. This projection is segmented to create several layers of elements. Note that all solid primitives are composed of 8-noded hexahedral elements and all shell primitives are composed of 4-noded quadrilateral elements. After you changed the mesh parameters simply press the *Apply* button to create the new mesh. To show the mesh in the Graphics View select the *View/Toggle Mesh lines* menu or use the '*m*' shortcut.

5.4.2 Editable Surfaces

An *editable surface* is an object that is defined via a surface mesh. The surface mesh can be edited via the tools on the *Edit panel*. If the mesh is closed and composed of triangles, the object can be meshed using the *Tetgen* tool, available from the *Mesh panel*. This tool will generate a tetrahedral mesh.

The surface of an editable surface can be modified via the tools on the *Edit panel*:

- **Project curve**: projects a curve on the mesh and modifies the mesh locally so that the curve becomes a crease edge of the mesh.
- **Partition**: Partition the mesh selection.
- **Smooth**: Smooths the surface mesh.
- **Edge collapse**: Collapses small edges. This tool can improve the quality of the mesh.
- **Decimate**: Remesh the triangular meshes using a Centroidal Voronoi Diagram based approach. Decimation is the process of decreasing the number of triangles in the mesh.
 - **Scale**: This option helps in creating a uniform triangular mesh. Scale value specifies the desired reduction in the total number of nodes in the final decimated mesh. For example a 0.4 value of scale will result in the final decimated mesh with 40% of current number of nodes.

- **Fix Mesh** – Provide options to fix specific problems in mesh
 - Remove duplicate elements.
 - Remove non-manifold elements.
 - Fix winding: This option works only for triangular mesh. This option checks if the nodes of the triangle are in same sequence clockwise/anti-clockwise direction for all the triangular shells.
 - Fill all holes: Finds and fills the holes in the triangular mesh by adding new triangular faces.

5.4.3 Editable Meshes

If the object is a so-called *editable mesh* the Mesh panel will list some mesh editing tools. Editable meshes don't have a geometry object associated with it so the mesh, or at least its surface, defines the geometry implicitly. This has some important consequences related to applying boundary conditions to an editable mesh. Any change to the mesh may also change the corresponding geometry and as a consequence any data that was associated with the previous geometry may become invalid.

Important Note. *It is best to first make the necessary modifications to an editable mesh before you apply any boundary conditions or loads. Any modifications to an editable mesh may invalidate boundary conditions and loads.*

Editable meshes can be edited on several levels, namely the *object level*, the *element level*, the *face level*, the *edge level*, and the *node level*. The element, face, edge, and node level are also referred to as the sub-object levels. When an object is an *editable mesh* (or editable surface), the Graphics control bar, will show additional buttons that allow you to select mesh items.



Enter the *element level*.



Enter the *face level*.



Enter the *edge level*.



Enter the *node level*.

When none of the sub-object levels are active, the *object level* is automatically active. If you are in one of the sub-object levels, you can return to the object level by deselecting the selected button on the selection tab or pressing the Esc button.

The Graphics control bar also provides several options that affect the way mesh items can be selected.

- *Select connected*: select all items that are connected to the selection. An angle criterion is used in addition to a connectivity criteria. The angle for this criterion can be set in the edit field next to this button.
- *Select via closest path*: This will select all items via a closest-path criterion between two selected points.
- *Select backfacing*: ignore items that are on the back of the mesh (and therefore not visible from the current viewing position).

To edit the geometry select the *Mesh* panel. Several tools will be displayed that allow you to modify the mesh. Most of these tools require a specific mesh selection mode to be active.

- **Auto Partition:** Partitions the surface, edges, and nodes of the mesh based on an angle criterion. If the Repartition elements is checked, the elements will be partitioned based on their connectivity.
- **Partition:** Partition the current mesh selection.
- **Smooth:** This option works only for triangular shell meshes. It smooths the mesh by iteratively moving points towards their neighbors and often results in better-shaped triangles and more evenly distributed nodes.
 - *Iterations*: Number of iterations to apply the smoothing
 - *lambda*: weight factor for scaling
 - *Preserve shape*: tries to preserve the overall shape of the object.
 - *Project*: project the smoothed nodes back to the original mesh
- **Discard Mesh:** Discards the volume mesh and only retains the surface as a shell mesh.
- **Mirror:** Mirror the mesh.
- **TetGen:** Replace the current mesh with a tetrahedral mesh.
- **Weld Nodes:** Weld the selected nodes together that are within a distance specified by the *threshold* edit field. Welding is useful to connect touching parts together. However, be aware that you might create unexpected errors in your geometry this way since the effects of welding are not always visible.
- **Refine Mesh:** Refines the triangular shell mesh by dividing each triangular face into 4 triangular faces. In this method a new node is added at the center of each edge of the triangle and new triangles are created using these new nodes.
- **Convert:** Convert the element type to a different type.
- **Add node:** Add a node to the model at the specified coordinates.
- **Invert:** Invert the selected elements.
- **PostBL:** Create a boundary layer from the selected surface faces.
- **Shell Thickness:** Set the shell thickness for the selected elements.
- **Set Fibers:** Set the fiber orientation of the selected elements.

Chapter 6

Materials

Materials are used to define the constitutive properties and behavior of the various parts in the model. This chapter explains how to add, edit, and assign materials to the different parts of your model.

6.1 Adding materials

Materials are added using the *Material Browser*. The Material Browser is accessed from the *Physics/Add Material* menu, using the *Ctrl+M* shortcut or by right-clicking on the *Materials* item in the Model Editor and selecting *Add Material* from the popup menu. The Material browser is displayed in Figure 6.1. The Material Browser allows you to compose a material. Note that it doesn't allow you to set the material properties. They will be set in Model Viewer.

In the first edit field you can enter a name for the material or leave it empty to accept a default name. The name of a material can also be changed later. Below the name field, a drop down list can be used to select the material category. The tree view below the category drop down box allows you to compose the material. When you select an item in this tree view, a drop-down box will appear next to it that allows you to choose a material type for the selected material component. Sometimes, selecting an option will create more material components. You do not need to select an option for all the components. You only need to set the ones that are relevant for your model. After you selected all the relevant components, click OK to create the material and add it to the model. The material will become selected in the Model Viewer.

6.2 Setting material parameters

After you create a material, you can set the material parameters in the Model Viewer. More specifically, when you select a material in the Model Viewer, the material properties are displayed in the Properties rollout. Note that materials can have child components, which can be selected by expanding the tree item. The components will have their own set of material parameters that can be altered.

Note that you can assign a color to each material. In the Object rollout on the Model Viewer, you will see a color selector next to the material's name. If you click on this you can change the color associated with that material. When assigning a material to a part (see below), the part will be shown using the material's color.

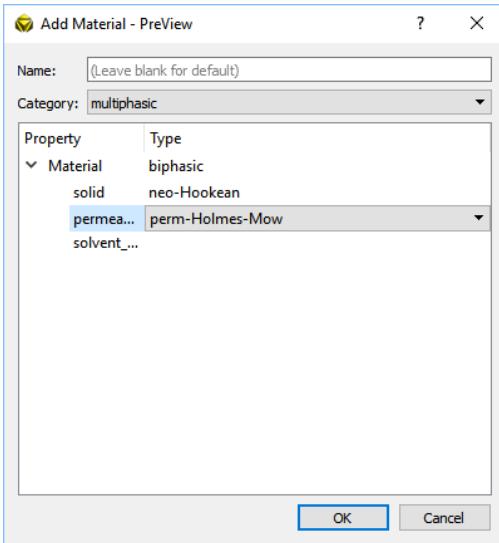


Figure 6.1: The Material Browser is used to add materials to the model.

6.3 Assigning materials

After you created a material and defined the material parameters you need to assign the material to your model. You can assign a material either to an object or to a part. To assign to an object, first make sure the object selection option on the toolbar is enabled (green circle). Next, select the object to which you wish to assign a material. Find the material in the Model Editor. Notice the selection panel in the Model Viewer. This panel contains a list that displays the parts to which this material is assigned. Initially, this list will be empty indicating that the material is not being used yet. When you now click on the + button, the selected object will be added to the list. More precisely, all the parts of the selected object will be added. To assign a material to only a part of an object, follow a similar procedure except now enable the part selection option on the toolbar (blue circle). Select the part in the Graphics View and then add the part by clicking on the + button on the material's selection list.

6.4 Creating a Solute Table

Solutes may be included in several types of materials in FEBio, including biphasic-solute, triphasic, and multiphasic materials. A global table of solutes is created by accessing the *Physics/Solute Table* menu. When adding materials that include solutes (Section 5.1), these may be selected from the solute table.

6.5 Creating a Solid-Bound Molecule Table

Solid-bound molecules (SBM) may be included in multiphasic analysis. A global table of SBMs is created by accessing the *Physics/Solid-Bound Molecule Table* menu. When adding materials that include SBMs, these may be selected from the SBM table.

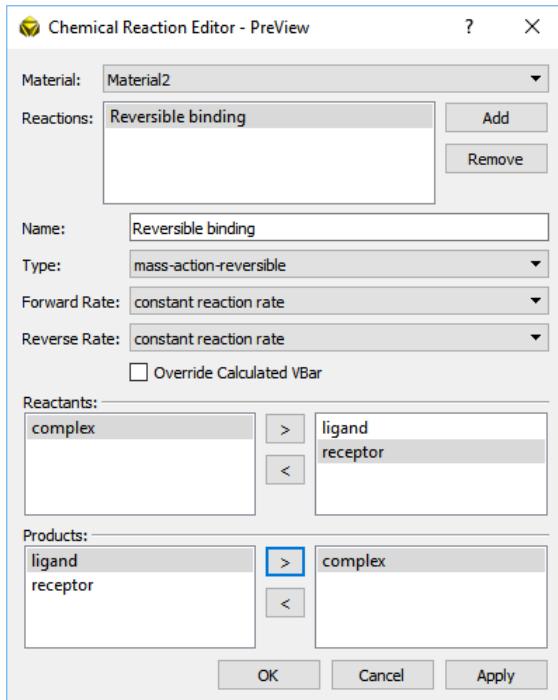


Figure 6.2: The Chemical Reaction Editor allows users to create and edit chemical reactions used in multiphasic analysis.

6.6 Adding Chemical Reactions

Chemical reactions may be included in some types of analysis (e.g. multiphasic analysis). Chemical reactions can only be added to previously created multiphasic materials as described in Section 6. Chemical reactions are defined between solutes (Section 6.4) or solid-bound molecules (Section 6.5). Only solutes and SBMs that have been included in the parent multiphasic material may be involved in a chemical reaction.

To create a chemical reaction, select the *Physics/Chemical Reaction Table* menu. The Chemical Reaction Editor will appear (Figure 6.2). First select the multiphasic material to which a chemical reaction will be added. Then, press the Add button, next to the reactions table. A new reaction will be added. The name of the reaction can be altered in the Name field. Select the type of reaction and choose the forward and reverse rates (the latter only if applicable.)

Under *Reactants* you will see two panels. The left panel shows all the solutes and sbms available in the model. The materials in red are not defined in the multiphasic material and should not be used to create a chemical reaction for this material. To include a species in the reaction move it to the right panel by pressing the > button between the two panels. You can remove a species from the reaction by pressing the < button. Repeat for all reactants of the chemical reaction. The *Products* panels work identically.

To create the chemical reaction and add it to the multiphasic material, simply press the OK button. Once a chemical reaction has been created, its material parameters may be set in the Model Viewer as described in Section 6.2. Multiple chemical reactions may be defined within the same multiphasic material. The Chemical Reactions Editor can also be used to modify or remove reactions from a material.

Chapter 7

Boundary Conditions and Loads

This chapter describes the various boundary conditions and loads that can be applied with Pre-View. These include fixed constraints, prescribed constraints, prescribed surface loads and tractions, body forces, etc.

7.1 Boundary Conditions

To apply a boundary condition, select the *Physics/Add Boundary Condition* menu. A dialog box appears that shows a list of available boundary conditions.

At the top of the dialog box the name of the boundary condition can be entered. Alternatively, this field can be left blank to accept a default name. Next, a drop-down list displays all the steps for which you can define a boundary condition. If you choose the *initial* step, the step will be applied in the initialization phase of the analysis and will remain active for all subsequent analysis steps. If you choose any other step, the boundary condition will only remain active during that step.

After you selected the step and the type of boundary condition, simply press OK to add the boundary condition to the model and edit the parameters in the Model Viewer.

In general, there are two types of boundary conditions. There are the *fixed constraints* and the *prescribed constraints*. For a fixed constraint, the corresponding degree of freedom is kept zero throughout the entire analysis. For a prescribed constraint, the value of the corresponding degree of freedom is defined through a load curve. You may wonder why the fixed constraints are available, since you can achieve the same result by defining a zero load curve for a prescribed constraint. The reason is that the degrees of freedom for fixed constraints are removed from the linear system of equations, reducing the computational time to solve the linear system. On the other hand, since the equations are removed, no reaction loads are calculated for fixed constraints. If you need to know for instance the reaction force on a boundary, you need to use a prescribed displacement even if the displacement is zero.

It is important to understand that with each prescribed constraint a load curve is associated. The actual value for the constraint at any given time is the product of the scale factor which you will enter in the properties dialog for the boundary condition and the value of the load curve at that time. Since by default the load curve will ramp from zero to one, the constraint value will ramp from zero to the specified value in a linear way. If you wish to modify the default curve you can edit it in the Curve Editor. See section 3.9 on details of dealing with load curves and the Curve Editor.

In the next sections we briefly discuss the available boundary conditions.

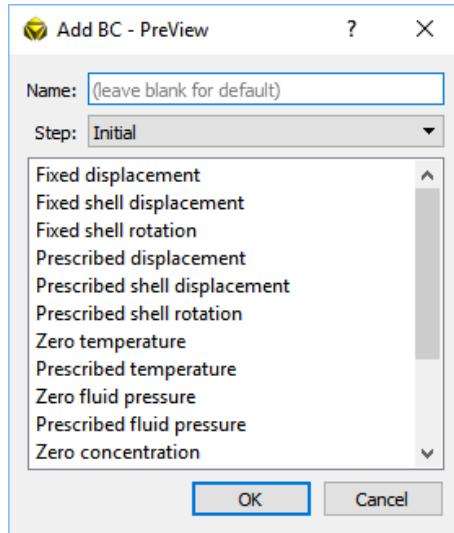


Figure 7.1: The Add Boundary Condition dialog box.

7.1.1 Fixed Displacement

A *Fixed Displacement* boundary condition allows you to fix a boundary. In the properties list, the x , y and z degrees of freedom are the translational degrees of freedom of the FE nodes.

7.1.2 Fixed Shell Displacement

Certain shell element formulations used in FEBio have two sets of displacement degrees of freedom. One for the front-face of the shell, and one for the back face. The usual displacement degrees of freedom are used for the front-face of the shell, and the shell displacement degrees of freedom are used for the back-face of the shell. This boundary condition allows you to fix the back-face shell degrees of freedom.

7.1.3 Fixed Shell Rotation

A *Fixed Rotation* allows you to fix the rotational degrees of freedom of a boundary. This is only used by certain types of shell elements.

7.1.4 Zero Fluid Pressure

This boundary condition can be used to set the fluid pressure to zero in biphasic analysis. A zero pressure boundary condition on a surface defines a free-draining surface, in other words, fluid will be able to leave the model through this surface.

This boundary condition differs from a zero prescribed fluid pressure (see below) in that the degrees of freedom for a zero fluid pressure boundary will be removed from the linear system.

7.1.5 Zero Temperature

This boundary condition can be used to set the temperature of a boundary to zero in a heat transfer analysis. It differs from a zero prescribed temperature in that the degrees of freedom for a zero

temperature boundary will be removed from the linear system.

7.1.6 Zero Solute Concentration

Use this boundary condition to set the solute concentration to zero for a multiphasic analysis (e.g., biphasic-solute, triphasic, etc.). It differs from a zero prescribed solute concentration in that the degrees of freedom for a zero solute concentration boundary will be removed from the linear system. Solutes must first be defined in the Solute Table.

7.1.7 Prescribed Displacement

A *Prescribed Displacement* allows you to define the translational degrees of freedom of a boundary. The value of displacement is the product of the associated load curve and the scale factor which is entered in the properties dialog box on the Model Viewer.

7.1.8 Prescribed Shell Rotation

This boundary condition allows you to prescribe the rotational degrees of freedom of a shell. This is only used by certain types of shell formulations.

7.1.9 Prescribed Shell Displacement

This allows you to prescribe the back-face shell displacement degrees of freedom. This is only used by certain shell formulations.

7.1.10 Prescribed Fluid Pressure

Use this boundary condition to prescribe the fluid pressure on a boundary for a biphasic analysis. The value of the prescribed pressure is the product of the associated load curve value and the scale factor which is entered in the properties dialog box.

7.1.11 Prescribed Temperature

Use this boundary condition to prescribe the temperature on a boundary in heat transfer analysis. The value of the prescribed temperature is the product of the associated load curve value and the scale factor which is defined in the properties dialog box.

7.1.12 Prescribed Solute Concentration

Use this boundary condition to prescribe the solute concentration on a boundary in multiphasic analysis (e.g., biphasic-solute, triphasic, etc.). The value of the prescribed solute concentration is the product of the associated load curve value and the scale factor which is defined in the properties dialog box. Solutes must first be defined in the Solute Table.

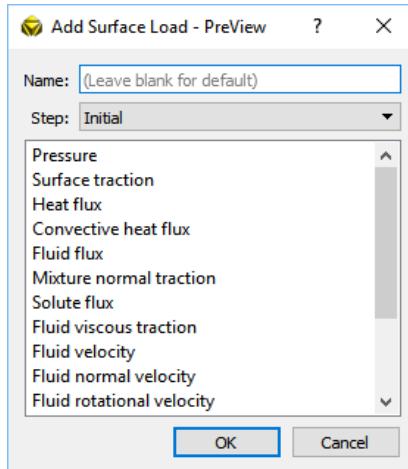


Figure 7.2: The Add Surface Load dialog box.

7.2 Surface Loads

Surface loads are applied in a similar way as boundary conditions. First select the items to which you wish to apply the load. Then, select the *Physics/Add Surface Load* from the menu bar. This opens up a dialog box from which you can select the available loads (Figure 7.2).

First, enter a name for the new surface load or leave the field empty to accept the default value. Then, select the step to which you wish to apply the load. Remember that applying a load to the initial step will cause the load to propagate through all the other steps, where if you select an analysis step, the load will only remain active during that step.

Next, select the type of load you wish to add from the list and click the OK button to confirm your choice. The load parameters can be entered in the properties box on the Model Viewer. The different available loads will be described in the following sections.

7.2.1 Pressure

The pressure load allows you to apply a pressure follower force to a surface. These pressure forces are always directed along the local surface normal and therefore change when the object undergoes a large deformation. Also note that, like prescribed constraints, a load curve is associated with the pressure force which by default will ramp the pressure value from zero to whatever value you entered.

7.2.2 Traction forces

Traction forces are similar to pressure forces but differ in two regards. First, the force vector is entered directly and does not need to be perpendicular to the surface. Second, the force remains constant and does not change with deformation. Note that a load curve is associated with the traction load amplitude which by default will ramp the amplitude from zero to whatever value you entered.

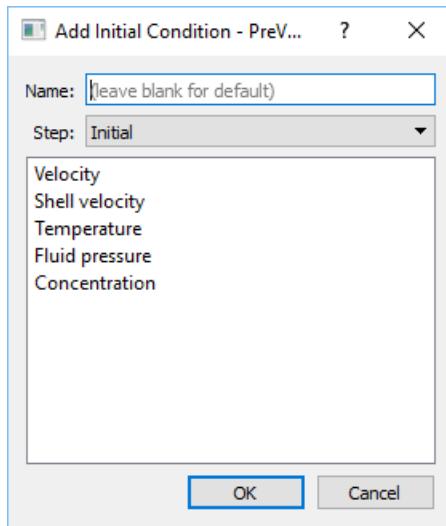


Figure 7.3: The Add Initial Condition dialog box allows users to create initial conditions for transient and dynamic analyses.

7.3 Initial Conditions

For dynamic problems, the user can define initial conditions in a similar way as boundary conditions. First, select the part, surface, edge or node to which you want to apply the initial condition. Then, from the Physics menu, select “Initial Conditions” to open the *Initial Conditions* dialog box. This dialog box presents a list of available initial conditions that can be applied to the current selection.

First, enter a name for the initial condition or accept the default. Then, select the step to which this initial condition needs to be applied. Next, select the initial condition from the list of available choices. The following sections describe the initial conditions in more detail.

7.3.1 Velocity

An initial velocity condition defines the velocity at the start of a simulation step. This condition will be applied to all the FE nodes of the current selection.

7.3.2 Shell Velocity

This allows you to set the initial velocity of a shell’s back face. This is only applicable for certain shell formulations.

7.3.3 Temperature

Use this to set the initial temperature of a transient heat transfer analysis.

7.3.4 Concentration

Use this initial condition to set the initial solute concentration for the current selection in a multiphasic analysis (biphasic-solute, triphasic, etc.). Solutes must first be defined in the Solute Table.

7.3.5 Initial Fluid Pressure

Use this initial condition to set the initial fluid pressure for the current selection in a biphasic or multiphasic analyses.

7.4 Assigning Boundary Conditions

To assign a boundary condition to a selection, first select the boundary condition (or boundary load, or initial condition) in the Model Viewer. Notice that a selection box appears below the tree view, which lists the model components to which this boundary condition is applied. Initially, this list may be empty. This list can be edited following the procedure detailed in section 3.7.1.

Chapter 8

Contact and Constraints

8.1 Rigid Body Constraints

The degrees of freedom of rigid bodies can be constrained in the same way as any other degree of freedom in the model. Before you can apply a constraint to a rigid body, you need to define a rigid body. In PreView, rigid bodies are implicitly defined when you create a rigid body material. A rigid body constraint can now be applied using the *Physics/Add Rigid Constraint* menu. It is important to note that when a rigid body is created none of its degrees of freedom are initially constrained.

In the *Add Rigid Constraint* dialog box first select the step in which this constraint will be active (or the *Initial* step if the constraint is to be active in all steps). Next, select the rigid body to which this constraint will be applied. Then, select a constraint from the list of available constraints that can be applied to a rigid body degree of freedom. The *Fixed Displacement/Rotation* constraint fixes the degree of freedom to zero. This implies that the rigid body will not be able to move or rotate in that degree of freedom. The *Prescribe Displacement/Rotation* lets you prescribe the value of the degree of freedom over time. The *Prescribe Force/Torque* constraint lets you apply a force or a torque (with respect to the rigid body's center of mass) to a rigid degree of freedom. Note that the prescribed displacements and force constraints have a load curve associated with them that can be edited in the Curve Editor.

8.2 Contact

PreView can be used to set up several types of contact conditions. These contact conditions allow the connection of non-conforming meshes to each other or the specification of non-penetration constraints. PreView supports the following contact interface categories.

Interface	Description
Rigid	Connect a deformable mesh to a rigid body
Sliding	Enforce a non-penetration constraint between two bodies
Tied	Tie two non-conforming surfaces together

To add a contact definition, select the *Physics/Add Contact* menu. A dialog box shows up that allows you to select the step for which the contact definition is to be active (or select *Initial* if the definition is to be active during all steps). To add a particular contact condition, select an option from the list and click on the *Add* button.

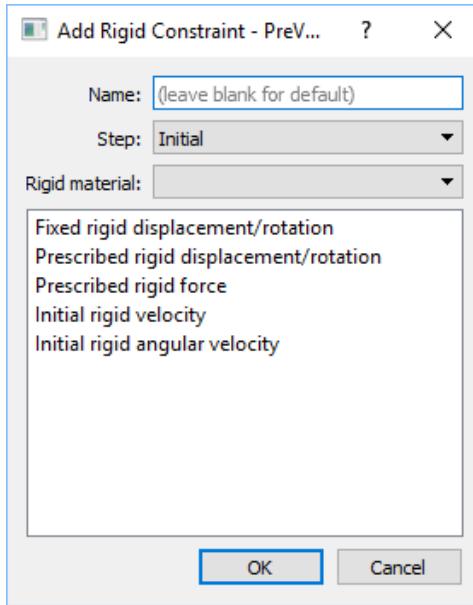


Figure 8.1: The Add Rigid Constraint dialog box allows users to constrain the degrees of freedom of a rigid body.

In FEBio, contact constraints are enforced using an *Augmented Lagrangian* approach. This implies that the Lagrange multipliers are only approximated to a user-specified tolerance. The following parameters will appear in nearly all contact interfaces.

- **augmented Lagrangian:** Turn the augmented Lagrangian method on or off. When off, a penalty method is used for constraint enforcement.
- **augmentation tolerance:** Set the convergence tolerance for the Lagrange multipliers.
- **Penalty factor:** The penalty factor controls the rate of convergence. A high penalty factor will try to reach the tolerance quickly, but if chosen too high might introduce instability into the system. If it is too low, convergence to within the specified tolerance might not be reached.

For sliding interfaces, there will usually be an option for *two-pass* option. In a usual contact implementation, the required integrations are only performed over one of the surfaces, usually referred to as the *slave* surface. If the contacting surfaces are perfectly smooth, it does not matter which surface acts as the slave or master surface. However, in an FE simulation the surfaces are discretized and are most likely non-conforming. Thus, the choice of master and slave surfaces is important and will introduce bias in the solution. It is usually advisable to select the more finely meshed surface as the slave. In the *two pass algorithm*, an attempt is made to reduce the bias by performing the contact calculations twice, with the roles of slave and master surface switched for the second pass. Although it may appear that the two pass algorithm is always the best choice, this is not always so. Certain contact applications perform better using a single pass. See the *FEBio Theory Manual* for a more detailed description of the contact model.

After the contact parameters are entered the user needs to define the *master* and *slave* surfaces. To do this, first close the contact interface dialog and select the contact interface in the Model Editor. In the Model Editor you will now notice two selection boxes, one for the master surface and one for the slave surface. The boxes work similarly as for boundary conditions. For

instance, to add a surface of your model to the master surface, select the surface in the Graphics View and press the '+' button in the master's surface selection box.

A description of the different contact definitions follows, although you should consult the FEBio User's Manual for a detailed overview of the parameters and use of the various contact interfaces.

8.2.1 Rigid Interfaces

There are two ways to make a deformable object interact with a rigid body, namely via a Rigid interface, or via a Rigid wall.

Rigid interfaces are used to connect a part of the deformable mesh to a non-conforming rigid body. By "non-conforming", we mean that the nodes comprising the mesh of the deformable object do not mate directly to the nodes comprising the rigid body at the interface. In other words, no nodes are shared between the two objects. The rigid interface requires only one parameter: the rigid material that defines the rigid body.

A rigid wall interface is similar to a sliding interface except that the master surface is a rigid plane. The geometry of the plane is simplified since the master surface is implicitly defined by the plane equation of the wall and does not need to be modeled explicitly. The plane can also be assigned a prescribed displacement in the Curve Editor. Just look for the name of the rigid wall and edit the *displacement* curve. In this case the plane equation generalizes to:

$$ax + by + cz + d + \varepsilon(t) = 0$$

where $\varepsilon(t)$ is the plane offset defined by the loadcurve.

After you select OK the rigid wall will be displayed in the Graphics View. If you don't see it, or if you wish to hide the rigid wall, you can set the appropriate option in the PreView Options dialog box. This dialog box can be accessed by pressing F6 or from the Tools/Options menu. In the dialog box, select the *Physics* options from the list on the left side and then check (or uncheck if you wish to hide the walls) the *show rigid walls* option.

8.2.2 Sliding Interfaces

A sliding interface defines a non-penetration constraint between two surfaces. This means that the surfaces are allowed to separate and slide across each other, but are not allowed to penetrate each other. If the contacting parts are biphasic or multiphasic, specialized contact interfaces can be used to allow fluid or solvent to flow across the contact interface when the parts are in contact. The following sliding interfaces are available.

Sliding node-on-facet This is a simple sliding interface that uses a nodal integration rule. Although this interface may be somewhat faster than the other, it is also the least accurate.

Sliding facet-on-facet. This sliding interface is similar to the *sliding node-on-facet* but uses a more accurate integration rule for evaluating the contact forces.

Sliding-elastic. A sliding interface that uses a different algorithm for detecting contact and evaluating the contact integrals than the *sliding node-on-facet* and *sliding facet-on-facet*.

Biphasic contact. This contact interface is similar to *sliding-elastic* but allows the fluid to flow across the contact interface for a biphasic analysis.

Biphasic-solute contact. This contact interface is similar to *sliding-elastic* but allows the fluid and the solute to flow across the contact interface for a biphasic-solute analysis.

Multiphasic contact. This contact interface is similar to *sliding-elastic* but allows the fluid and solutes to flow across the contact interface for a biphasic analysis.

Symmetry Plane. Defines a symmetry plane for emulating axisymmetric analysis.

8.2.3 Tied Interfaces

Tied interfaces can be used to tie two non-conforming surfaces together. It is also possible to make the tie temporary or define a tie-breaking criterion. Special versions of tied interfaces are available for biphasic and multiphasic analysis that allow the fluid and solvents to cross the interface.

Tied node-on-facet. Similar to the *sliding node-on-facet*, but surfaces are not allowed to slide across or break away from each other.

Tied facet-on-facet. Similar to *tied node-on-facet* but a more accurate surface integration is performed to evaluate the contact forces.

Sticky. Identical to *tied node-on-facet* but users can define a criterion for tie-breaking.

Periodic boundary. This type of tied interface will allow you to force the displacement between two surfaces to be identical.

Tied biphasic contact. A special tied interface that allows the fluid in a biphasic analysis to flow across the contact interface.

Tied multiphasic contact. A special tied interface that allows the fluid and solvents in a multiphasic analysis to flow across the contact interface.

Chapter 9

Defining Analysis Steps

The actual analysis that is to be performed is defined through *steps*. A model may define as many steps as needed, and each step defines what is to be calculated for that particular step.

9.1 The Initial Step

When you start PreView or create a new model using the *File/New* menu, you'll notice that PreView by default creates one step, namely the *Initial* step. This initial step does not perform an actual analysis; instead it is a placeholder for the initialization phase of the analysis. Currently, the most important aspect of this step is that all boundary conditions, loads, constraints, etc. that are defined in this step will remain active during all subsequent analysis steps. This is different from an analysis step in that e.g. a boundary condition defined in an analysis step will only remain active during that step.

9.2 Adding an Analysis Step

An analysis step is added using the *Physics/Add Step* menu. Alternatively, you can also add a step by right-clicking on the *Steps* item in the Model Editor and select *Add* from the popup menu. A dialog box appears where you can now set the Step settings.

The following analysis steps are available:

- *Structural Mechanics*: define a quasi-static or dynamic, large deformation, structural mechanics analysis
- *Heat Transfer*: steady-state or transient heat transfer analysis.
-
- *Biphasic*: solve a coupled nonlinear transient or steady-state biphasic (solid+fluid) analysis.
- *Biphasic-solutes*: solve a coupled nonlinear transient or steady-state biphasic analysis with a single solute.
- *Multiphasic/Solutes*: solve a coupled nonlinear transient or steady-state multiphasic that may have an unlimited number of solutes, solid-bound molecules, and chemical reactions.
- *Fluid Mechanics*: Define a dynamic fluid mechanics analysis.

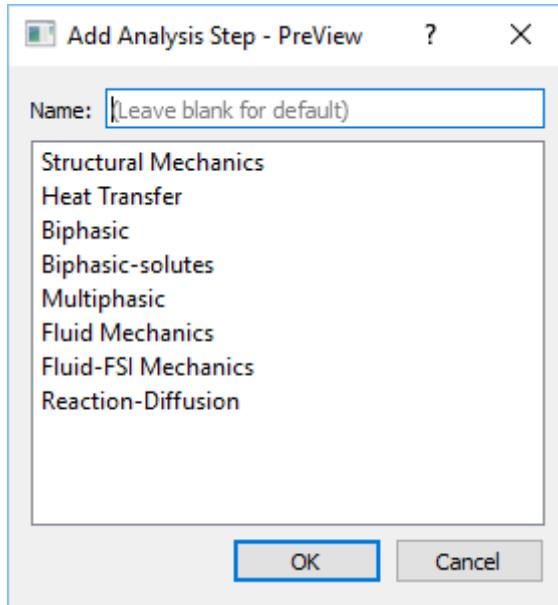


Figure 9.1: The Add Analysis Step allows users to create steps for their model.

- *Reaction-Diffusion*: Define a transient reaction diffusion analysis.

After you select an analysis type and click OK the analysis parameters can be edited in the Model Viewer. These settings are discussed next.

9.2.1 Analysis

The analysis settings allow you to choose the specific analysis type. This usually means steady-state or transient/dynamic.

9.2.2 Time Settings

These settings control the time step controller. The solution can be progressed using fixed time steps or adaptive time steps, depending on whether you check the *use auto time stepper* option. When checked, it activates the *auto timestep controller* which will adapt the step size based on convergence information. The following parameters control the auto time step controller

- *Timesteps*: define the number of time steps to solve for. If the *auto timestep controller* is activated, this is merely a suggestion. The actual number of timesteps taken will vary based on the parameters below.
- *Step size*: defines the initial timestep size that will be taken by the time controller. If *auto timestep controller* is activated, the actual time step size taken will vary based on the parameters below. The product of the “Timesteps” parameter and the “Stepsize” parameter determines the termination time.
- *Max retries*: the maximum number of times that a particular timestep will be retried.
- *Optimal iterations*: The expected average number of iterations required to converge a single time step.

- *Min step*: the minimum stepsize that may be taken by the auto timestep controller.
- *Max step*: the maximum stepsize that may be taken by the auto time step controller. If a *must point curve* is specified this value will be overridden by the value of this curve.
- *Cutback*: select the method for reducing the step size if an iteration fails.

The *use must-points* option allows to use so-called must-points. The must-points are time points, defined through a load curve, for which the solution will be forced to pass through. The *must point curve* is edited in the Curve Editor. The value of the must point curve defines the maximum time step size that can be taken up to that point. If a must point curve is defined, the *Max step* value in the Project Settings is ignored.

9.2.3 Nonlinear Solver

Set the options for the nonlinear Newton solver which is used for the nonlinear analysis types (all analyses except the heat transfer analysis use this solver). The following options can be set.

- *Displacement tolerance*: convergence tolerance on displacement norm
- *Energy tolerance*: convergence tolerance on energy norm
- *Residual tolerance*: convergence tolerance on residual norm
- *Line search tolerance*: The convergence tolerance for the line search.
- *Minimal residual*: If the residual norm drops below this value, the time step will automatically converge.
- *Quasi-Newton method*: Select the particular variation of Newton's method that will be used by the nonlinear solver.
- *Max reformations*: Max number of stiffness reformations per time step
- *Max BFGS updates*: Max number of BFGS iterations between reformations
- *Reform on diverge*: When checked, the global stiffness matrix will be reformed if an iteration diverges.
- *Reform each timestep*: When checked, the global stiffness matrix will be reformed at the beginning of each step. For linear or slightly nonlinear problems, turning this option off may result in significant performance improvements.

9.2.4 Linear Solver

In any (implicit) finite element code like FEBio, the linear solver does most of the work. The type of linear equation that has to be solved depends on the type of analysis, but some of the settings can be user-controlled. The following settings can be changed.

- *Matrix storage*: Some algorithms have both a symmetric form or an unsymmetric form. This option will set the preferred matrix format.

Chapter 10

Running FEBio from PreView

FEBio can be called from within PreView by selecting the *Tools\Run FEBio* menu, or from the main toolbar by clicking the corresponding tool button . The Run FEBio dialog box shows up (Figure 45) where users can set the following fields.

- **FEBio path:** The path to the FEBio executable. If FEBio is setup to run from anywhere on the command line, you can just enter the FEBio command here. If not, you may need to specify the full path to the FEBio executable here.
- **Working directory:** Specify the location where PreView will store the FEBio input file.
- **Filename:** Enter the name of the FEBio input file here. PreView will automatically export the current project to this file before running FEBio.
- **Debug mode:** Check this to run FEBio in debug mode.

By pressing OK, PreView will first export the current project to the file and location as specified, then call FEBio. The FEBio output will be shown in the log panel. If this panel is not visible, you can activate it from the *View\Log* menu.

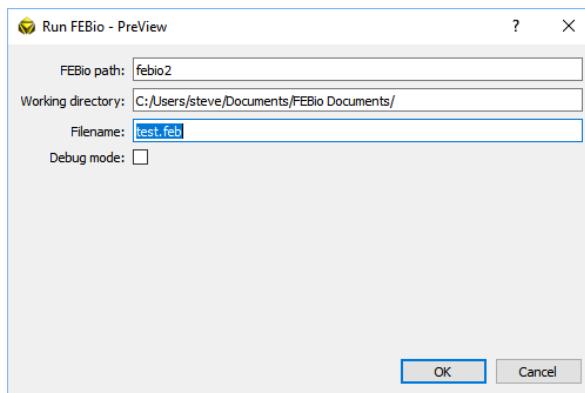


Figure 10.1: The Run FEBio dialog box lets you call FEBio from within PreView.

Chapter 11

Mesh Import Formats

This appendix details the different file formats that are supported by PreView. If you don't find your preferred format (or keywords) in this list, you may request them by posting a feature request on the PreView forum. Please keep in mind that we can only support publicly available text based formats.

11.1 FEBio

FEBio is a nonlinear finite element solver designed specifically for computational biomechanics problems. PreView was designed to create FEBio input files and supports most of the FEBio features. See the FEBio User's Manual and Theory Manual for a detailed description of FEBio's features.

11.2 NIKE3D

NIKE3D is a nonlinear finite element solver designed by Lawrence Livermore National Laboratories (LLNL). The following sections are supported:

- Control section
- Material deck: materials 1, 15, 18, 20 and 63 are supported.
- Node point deck
- Hexahedron deck
- Shell element deck
- Rigid node and facet deck
- Sliding surface deck: contact types 2 and 3 are supported.
- Stonewall and symmetry plane deck
- Load curve deck
- Concentrated nodal loads deck

- Pressure boundary condition deck
- Displacement boundary condition deck
- Base acceleration Body Force Loads deck

11.3 HyperMesh ASCII

This is the hypermesh ASCII format. The following keywords are supported.

- node
- component
- tria3
- tetra4
- hexa8

11.4 ABAQUS

The following ABAQUS®keywords are supported in PreView.

- ELEMENT
- ELSET
- END PART
- END INSTANCE
- HEADING
- INCLUDE
- INSTANCE
- MATERIAL
- NFILL
- NGEN
- NODE
- NSET
- PART
- SURFACE
- SURFACE INTERACTION

The following element types are recognized.

- C3D8
- C3D6
- C3D4
- C3D10
- R3D4
- S4
- S3
- S3R
- R3DR

11.5 LSDYNA keyword

The following LSDYNA®keywords are supported.

- ELEMENTSHELL
- ELEMENTSHELLTHICKNESS
- ELEMENTSOLID
- END
- MATELASTIC
- NODE
- PART
- SETSEGMENTTITLE

11.6 ANSYS

The following ANSYS®keywords are supported.

- EBLOCK
- NBLOCK

11.7 DXF

Currently, PreView only reads in *polylines*, described by vertices and triangulated faces.

11.8 Hypersurface ASCII

The following keywords are supported.

- Vertices
- Triangles

11.9 GMsh

The following GMsh keywords are supported.

- MeshFormat
- PhysicalNames
- Nodes
- Elements

11.10 BYU format

The BYU format, developed at Brigham Young University, is used to describe a surface mesh. All features of this format are supported.

11.11 VTK format

The legacy VTK format is provided by the Visualisation Toolkit. Preview only supports the ASCII format. The following keywords are supported:

- DATASET- POLYDATA, UNSTRUCTUREDGRID
- POINTS
- POLYGONS / CELLS
- POINTDATA
 - SCALARS – ShellThickness
 - SCALARS – ScalarData
- CELLTYPES

Currently Preview only supports the triangular and quadrilateral shell mesh and hexahedral volume mesh.