Slicer3 Training Tutorial
IA-FEMesh v1.0
Proximal Phalanx Bone

MIMX Laboratory

Center for Computer Aided Design

Engineering Research Facility

The University of Iowa

Iowa City, IA

Learning Objectives

Following this tutorial, you'll be able to:

1) find the IA-FEMesh module in Slicer3

2) generate a mesh representation of the proximal phalanx bone of the human hand

3) apply IA-FEMesh to your own application

Prerequisites

This tutorial assumes that you have already completed the tutorial **Data Loading and Visualization**. Tutorials for **Slicer3** are available at the following location:

• **Slicer3** tutorials
http://www.na-mic.org/Wiki/index.php/Slicer3.2:Training

Materials

3DSlicer

This tutorial requires the installation of the **Slicer3** software and the tutorial dataset. They are available at the following locations:

• **Slicer3** download page (*Slicer 3.3 alpha after 12/30/2008*)
http://www.slicer.org/pages/Downloads/

• IA-FEMesh dataset
http://www.na-mic.org/Wiki/images/5/5a/MeshTutorialExampleData.zip

Disclaimer: *It is the responsibility of the user of Slicer to comply with both the terms of the license and with the applicable laws, regulations, and rules.*
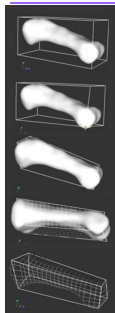
Overview

3DSlicer

• **Part A:** Introduction to the IA-FEMesh module

• **Part B:** Mesh the proximal phalanx bone

3DSlicer

Part A:
IA-FEMesh Module in Slicer3

Locating IA-FEMesh

1. Open Slicer3
2. Locate the "Modules" drop down menu
3. Select **IA-FEMesh** in the "Modules" drop down menu

Part B:
Meshing the proximal phalanx bone

Loading Image Files

1. Select the **Image Tab**
2. Select **Load** on drop down menu
3. Select the image file to load
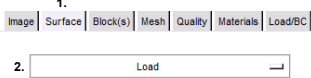4. Uncheck the checkbox in the Object Manager to temporarily hide the loaded image

Loading Surface Files

1. Select the **Surface Tab**
2. Select **Load** on drop down menu
3. Select the surface file to load

1.

| Image | Surface | Block(s) | Mesh | Quality | Materials | Load/BC |

2. | Load | ↵ |

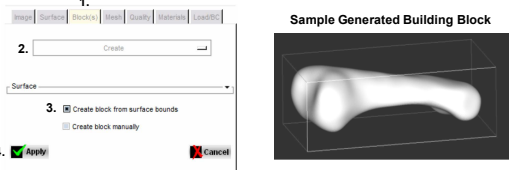## Creating a Building Block

1. Select the **Block(s)** Tab
2. Select **Create** from the drop down menu
3. Check **Create Block from Surface Bounds**
4. Click **Apply**

1.

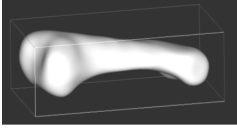| Image | Surface | Block(s) | Mesh | Quality | Materials | Load/BC |

2. | Create | ↵ |

Surface | ▼ |

3. ☒ Create block from surface bounds
☐ Create block manually

4. ✓ Apply   ✗ Cancel

Sample Generated Building Block

Manipulating the Building Block

1. Select the **Blocks** tab
2. Select **Build/Edit** from the drop down menu
3. Select the 🔽 from the building blocks toolbar
   a) Allows manipulation of vertices, edges, and faces of building blocks
   b) Red spheres will appear at the vertices. The size of these spheres can be scaled

1.

| Image | Surface | Block(s) | Mesh | Quality | Materials | Load/BC |

Create
Load
✓ Build/Edit
Delete
Save

2. | Build/Edit |

3a.

3b. Scale Spheres: |

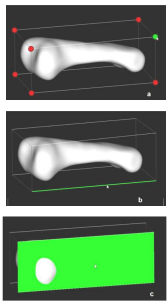## Manipulating the Building Block(2)

4. To move a vertex or face, simply click the **left mouse button** and drag it to the desired position
5. To move an edge, click the **middle mouse button** and drag it to the desired position
6. Click **Cancel** to exit building block manipulation
7. To save the building block, select **Save** from the drop down menu

- Note:
  - Active elements turn green
  - Inactive elements remain red
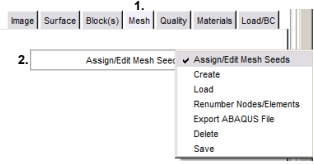

Assigning Mesh Seeds to the Building Block

1. Select the **Mesh** tab
2. Select **Assign/Edit Mesh Seeds** from the drop down menu

1.

| Image | Surface | Block(s) | Mesh | Quality | Materials | Load/BC |

2. Assign/Edit Mesh Seeds
- ✓ Assign/Edit Mesh Seeds
- Create
- Load
- Renumber Nodes/Elements
- Export ABAQUS File
- Delete
- Save
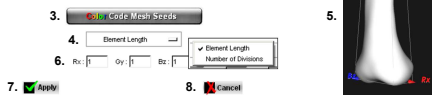

Assigning Mesh Seeds to the Building Block(2)

3. Select the **Color Code Mesh Seeds** button to visualize the distribution of mesh seeds
4. To change this assignment, select either **Element Length** or **Number of Divisions** from the drop down menu.
5. Click on a block with the left mouse button
6. Enter the values of your choice
7. Select **Apply**
8. Select **Cancel** to exit the mesh seeds operation

3. Color Code Mesh Seeds

4. Element Length
- ✓ Element Length
- Number of Divisions

5.

6. Rx: 1   Gy: 1   Bz: 1

7. ✓ Apply   **8.** ✗ Cancel

5

Creating a Mesh

1. Select the **Mesh** tab
2. Select **Create** from the drop down menu
3. Select **Volumetric Mesh**
4. Select **Building Block** in the following drop down menu
5. Provide starting **node/element numbers** and a **descriptive label** to be associated with the mesh definition
6. Selection **Elliptical** in the Interpolation drop down menu
7. Uncheck **Perform Smooth** for the initial attempt
8. Select **Apply** to generate the mesh

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-16-

Checking Mesh Quality

1. Select the **Quality** tab
2. Select **Evaluate/Display Mesh Quality** from the drop down menu

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-17-

Checking Mesh Quality (2)

3. Select your **Metric of Choice** from the Metric drop down menu
4. Click the **Summary Report** button
5. Close summary report window after viewing
6. Click **Cancel** to exit the mesh quality check module

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-18-

Improving Mesh Quality

1. Select the **Quality** tab
2. Select **Mesh Improvement** from the drop down menu
3. Check to ensure that the surface, block, and mesh of interest populate the **Mesh Component** frame
4. Select an **interpolation method** from the Interpolation drop down menu
5. Enter the desired **number of smoothing iterations** for the external nodes
6. Click **Apply**

Improving Mesh Quality(2)

7. Adjust the smoothing parameters by toggling between the **Evaluate/Display Mesh Quality** and **Mesh Improvement** operations until the desired mesh is achieved
8. While displaying the metric of interest in the Evaluate/Display Mesh Quality module, press the 🎚️ button to invoke a cutting plane to view the internal elements.
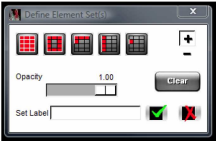
Assigning Material Properties – User Defined

1. Select the **Materials** tab
2. Select the **User-Defined** option from the main menu
3. To add additional element sets, press Element Set Definitions
   • A number of options are available for selecting element sets
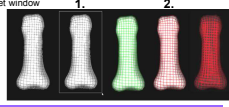
Assigning Material Properties – User Defined(2)

Element Set Definition Examples
- Cortical Bone Set Definition
  1. Select the surface element button: ▣ .
  2. Hold **Ctrl button** while using the **left mouse button** to drag a rubberband box around the elements of interest
     - Selected elements will be green
  3. To accept the chosen elements, click the **right mouse button** while hovering over the mesh
     - Accepted elements will turn red
  4. **Opacity** can be modified to better visualize the data
  5. Once the selection is finalized, enter a **Set Label** (e.g. cortical bone)
  6. Click ✔ to accept the selection
- Cancellous Bone Definition
  1. Repeat the steps above except hit the ━ button prior to accepting the element selection and assigning a new Set Label.
  2. Reduce the **opacity** to see the element set you have defined
  3. Click the ✖ button to close the Define Element Set window

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-22-
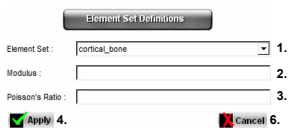

Assigning Material Properties – User Defined(3)

- Material Property Assignments
  1. Select an element set using the "Element Set" drop down menu
  2. Enter the desired **Modulus**
  3. Enter the desired **Poissson's Ratio**
  4. Click **Apply**
  5. Repeat this procedure for each material assignment
  6. Click **Cancel** to exit the operation

Element Set Definitions

Element Set : cortical_bone   **1.**
Modulus :   **2.**
Poisson's Ratio :   **3.**
✔ Apply **4.**   ✖ Cancel **6.**

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-23-
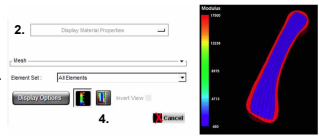

Assigning Material Properties – User Defined(4)

- Visualizing the Material Property Assignments
  1. Select the **Materials** tab
  2. Select **Display Material Properties** from the drop down menu
  3. Select the desired element set from the Element Set drop down menu
  4. Use the 🔳 button to display a cutting plane which may be manipulated to view internal element definitions

2. Display Material Properties
Mesh
3. Element Set : All Elements
Display Options   Invert View
4. ✖ Cancel

IA-FEMesh Tutorial - MIMX Laboratory
National Alliance for Medical Image Computing
-24-
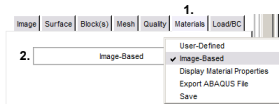
Assigning Material Properties – Image Based

1. At this point, you can overwrite your user-defined properties, or skip this step entirely
2. Select the **Materials** tab
3. Select the **Image-Based** option from the main menu
4. Expand the "Mesh & Image Assignments" frame
   - Ensure that the appropriate Mesh and image definitions are present

Assigning Material Properties – Image Based (2)

5. For each element set previously defined:
    a) Select the element set of interest
    b) Assign a Poisson's Ratio
    c) Assign Young's Modulus by selecting the [E=a+bp²] button and adjust the constants for the modulus calculation
    d) Select a method of calculating density for a given element
    e) Select **Apply in the Image-Based Elastic Modulus** window
    f) Select **Apply in the Materials Panel** to commit the image-based properties to the mesh
    g) Repeat this procedure for all element sets that were previously defined
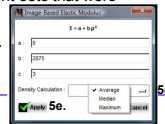
Assigning Load and Boundary Conditions

1. Select the Load/BC tab
2. Select the STEP – Load/BC Assignments from the drop down menu
3. Select [Node Set Definitions]
4. Select ▦ from the Node Set toolbar
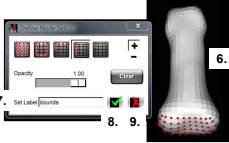   - This enables the nodes associated with a face of a building block to be readily chosen
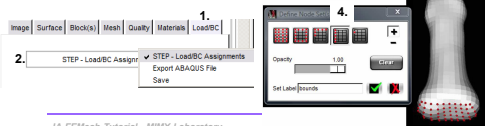
Assigning Load and Boundary Conditions (2)

5. Hold the **Ctrl button** and use the **left mouse button** to **choose a node** associated with the building block face of interest
   - Nodes associated with selected face will be highlighted in green
6. To accept the chosen nodes, click the **right mouse button** while **hovering over the mesh**
   - All nodes associated with the chosen face will turn red
   - Opacity can be changed using the slider bar
7. Once the selection is finalized, **enter a Set Label**
8. Click ✅
9. Click ❌
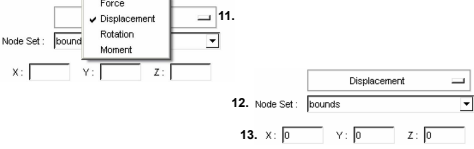
Assigning Load and Boundary Conditions (4)

10. Provide a **descriptive heading** in the Step Subheading textbox
11. Select desired **Load/Displacement** type in the drop down menu
12. Select the appropriate **Node Set**
13. Assign **x, y, and z directions**
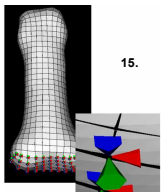
Assigning Load and Boundary Conditions (5)

14. Select Apply to update the Load/BC assignments
15. Visual confirmation will be provided on the mesh in the View Panel

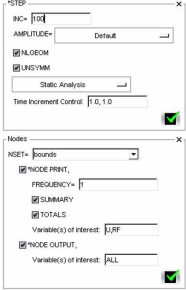## Assigning *STEP Definitions

1. Select the ***STEP Definitions** button
   - Please refer to the ABAQUS manual regarding these parameters
2. Use the **NSET** and **ELSET** drop down menus to select the sets of interest
3. Click **Apply** in each submenu to commit your selection

Final Steps

- Save the FE Mesh
  1. Select **Save** from the Mesh Tab
     - This can be performed at any step through the mesh development process

- Export the FE Mesh in ABAQUS file format
  1. Select **Export ABAQUS file** from the Mesh Tab
     - This can be performed at any step through the mesh development process

Acknowledgements