



NA-MIC

National Alliance for Medical Image Computing

<http://na-mic.org>

# Image to Finite Element Mesh: An End to End Workflow

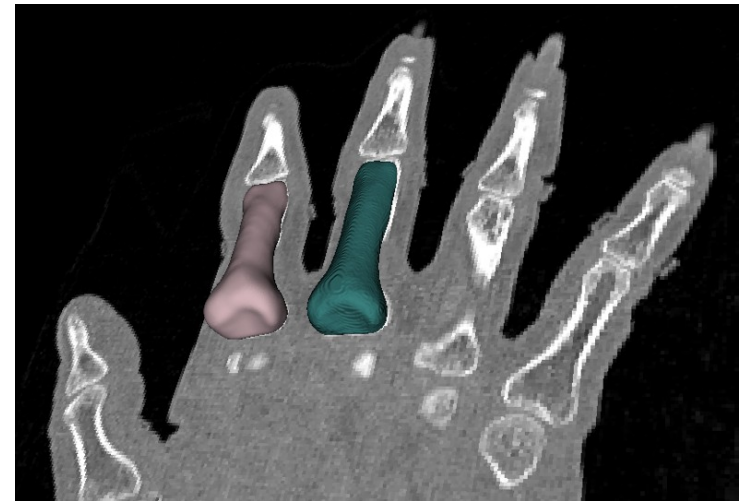
Nicole Grosland, Ph.D.

Vince Magnotta, Ph.D.

The University of Iowa

Steve Pieper, Ph.D., Isomics

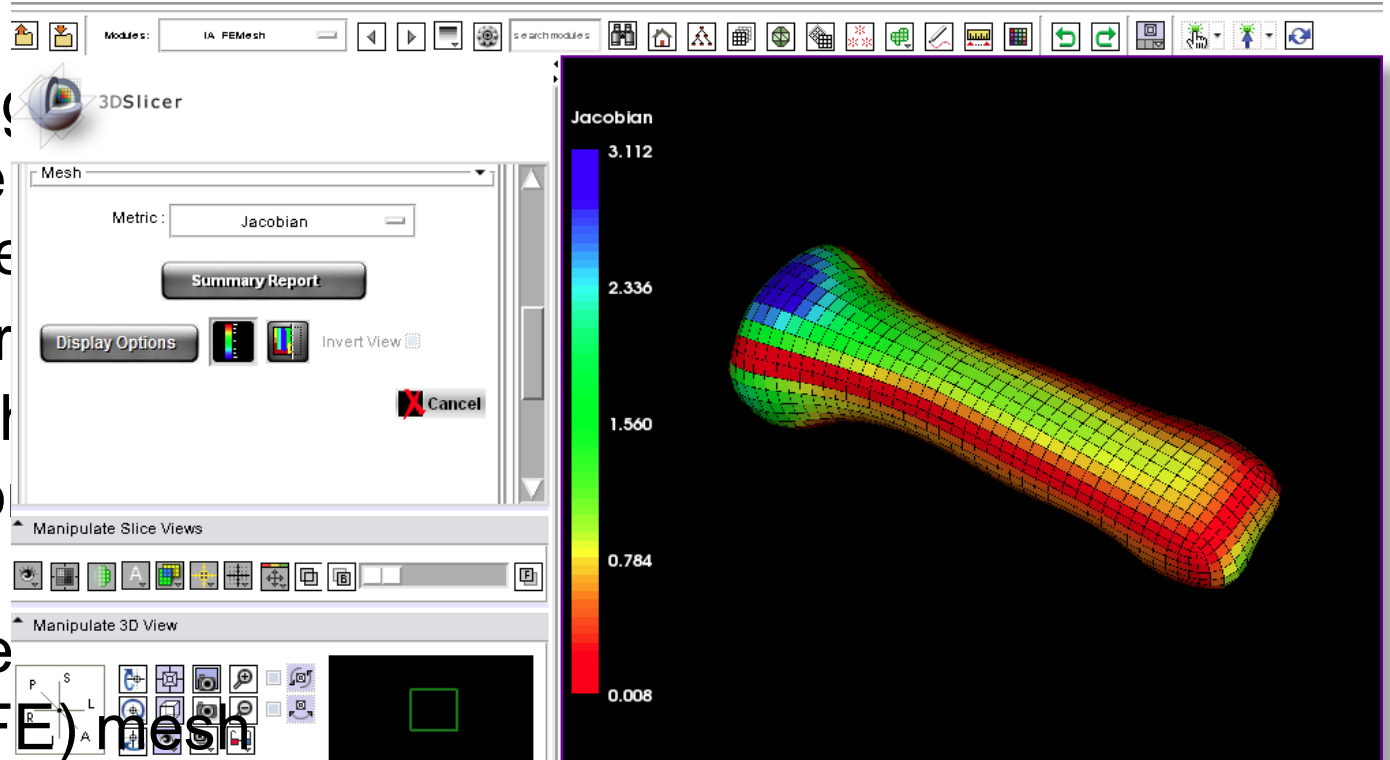
Curtis Lisle, Ph.D., KnowledgeVis





# Learning Objective

After following the workflow from image to the creation of a 3D volume element (FE) mesh, you will be able to use Slicer3 to evaluate the Jacobian of the mesh.





# Pre-requisites

---

- This tutorial assumes the user is familiar with loading image datasets and creating basic label maps in Slicer. Please consult the following prerequisite material if you want to brush up on these skills first:

- Data Loading & Visualization tutorial

- Sonia Pujol, Ph.D., Harvard/SPL
- [http://www.slicer.org/slicerWiki/images/c/c9/3DDataLoadingAndVisualization\\_Slicer3.6\\_SoniaPujol.pdf](http://www.slicer.org/slicerWiki/images/c/c9/3DDataLoadingAndVisualization_Slicer3.6_SoniaPujol.pdf)

- User<sup>f</sup> documentation for the Slicer3 Editor module

- Steve Pieper, Ph.D, Harvard/SPL, Isomics
- <http://www.slicer.org/slicerWiki/index.php/Modules:Editor-Documentation-3.6>



# Pre-requisites

---

- The user may use hand segmentation or any segmentation algorithm to create surfaces for meshing. One of the simplest algorithms to use is Slicer3's Fast Marching Segmentation module:
- Fast Marching Segmentation documentation
  - Adriy Fedorov, Ph.D., Harvard/SPL
  - <http://www.slicer.org/slicerWiki/index.php/Modules:FastMarchingSegmentation-Documentation-3.6>



# Material

---

- This tutorial requires the installation of the Slicer3.6 release and the tutorial dataset. They are available at the following

locations:

- Slicer3.6 download

<http://www.slicer.org/pages/Downloads/>

- Tutorial dataset:

–<http://www.slicer.org/slicerWiki/index.php/File:IAFEMeshData-TutorialContestSummer2010.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.



# Material

---

- A Users Guide is available for the IA-FEMesh module, which covers the step by step process of mesh creation.
- For additional assistance, consult the Users Guide available here

–[https://mri.radiology.uiowa.edu/downloads/IA-FEMesh\\_Manual\\_version1.pdf](https://mri.radiology.uiowa.edu/downloads/IA-FEMesh_Manual_version1.pdf)



# Material

---

- Interactive screencasts showing the following operations are available from the NA-MIC wiki website:
- Creating and importing a Slicer model
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-1-labelmap-model-import.mov>
- Simple Editing of Building Blocks around a surface
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-2-building-blocks.mov>
- Creating a mesh from a surface and building blocks
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-3-mesh-creation.mov>
- Evaluating and Improving Mesh Quality
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-4-mesh-quality-tab.mov>



# Platform

---

- The IA-FEMesh Slicer3 Module was developed using both Linux (32 and 64 bit) and Mac (OS X 10.4 - 10.6)
- The module has been tested on both Linux 64-bit and Mac (OS X 10.6)





# Overview

---

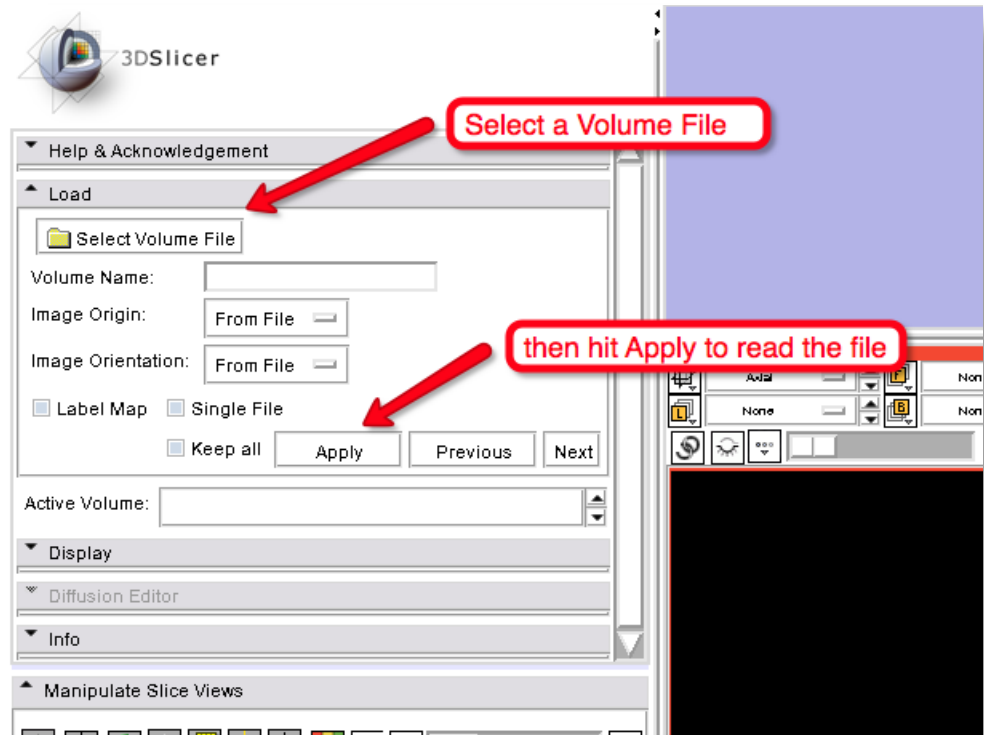
In this tutorial, we will...

- Create a label map and a corresponding surface model of a structure of interest from a source image
- Import the surface model into the IA-FEMesh module
- Interactively create a volumetric hexahedral mesh of the object of interest
- Examine and improve the quality of the mesh
- Export the mesh for continued processing in a commercial finite element solver (e.g. Abaqus)
- Review the steps to create an object mesh from imagery source



# Loading Image Data

1. Select the Slicer3 Volume Module
2. Select the “Volume File” button to choose a file from the filesystem. Many image formats are supported.
3. Hit “Apply” to read the image into the current session





# Creating a Labelmap

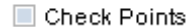
---

- A label map is a special type of volume where each voxel contains a code indicating the object to which the voxel belongs
- Label maps are loaded and saved like images, but handled differently by Slicer because of their purposes as indicators
- At right, we see a label map indicating the presence of the finger bone superimposed over corresponding background voxels from a CT image of the finger





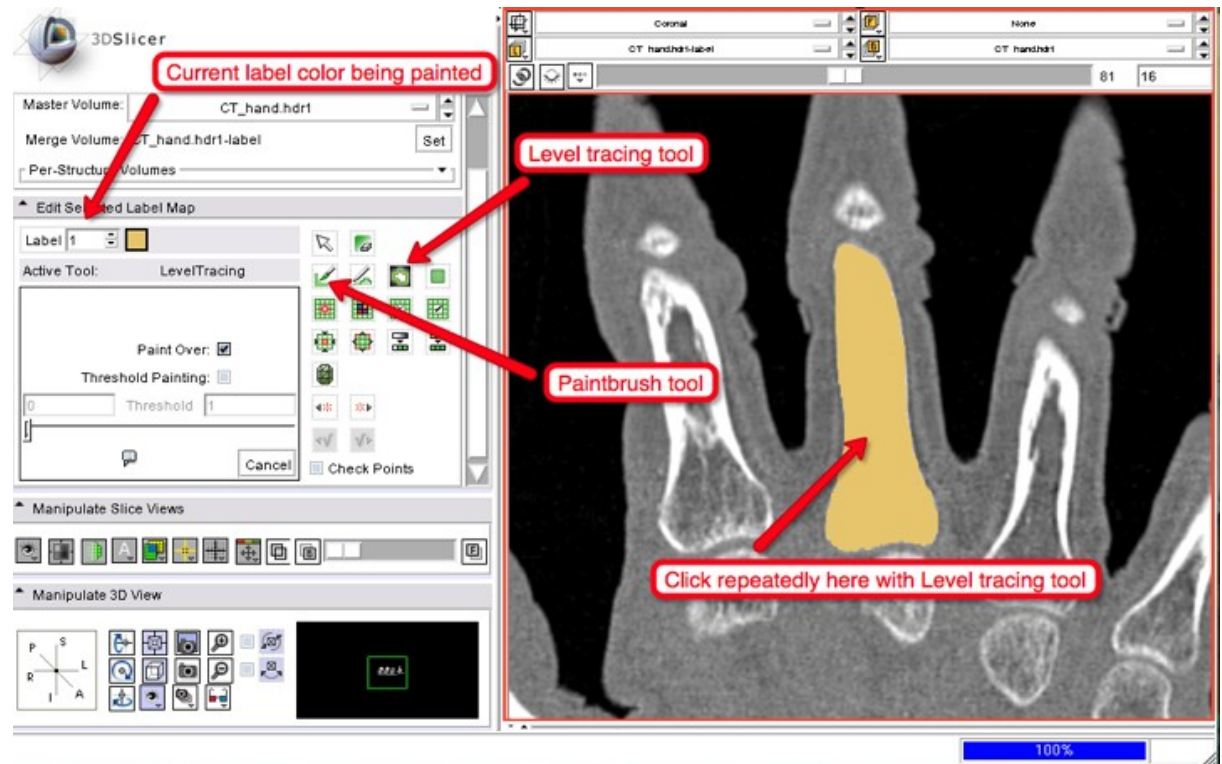
- The paintbrush is a way to clean up or add missing voxels after Level Tracing





# Creating a Label map

- Select the Level Tracing tool, move it around on the image, and click repeatedly to select the desired region
- Repeat the voxel selection operations for each slice containing the object to be segmented





# More about Segmentation

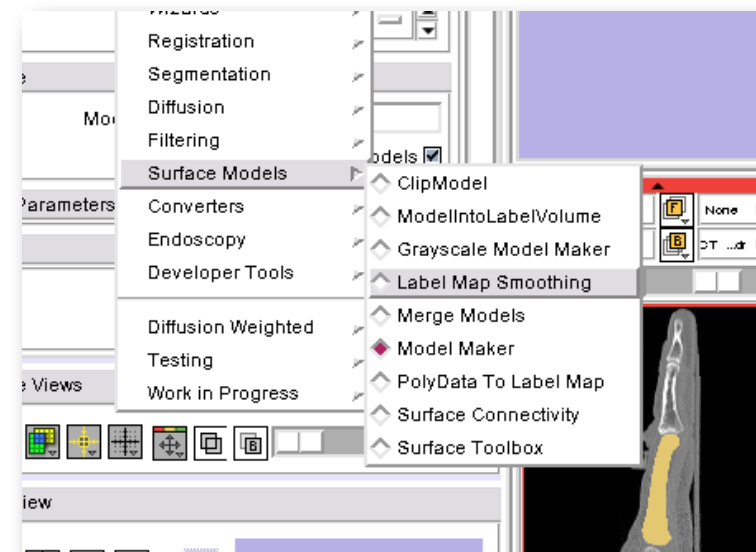
---

- Clearing unwanted voxels can be accomplished by painting with the clear label color (label=0)
- Step-by-step instructions for using the Slicer3 Editor are available on the Slicer wiki site:
  - <http://www.slicer.org/slicerWiki/index.php/Modules:Editor-Documentation-3.6>
- A screencast is available which demonstrates label maps and shows how models are created:
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-1-labelmap-model-import.mov>



# Smoothing the Label

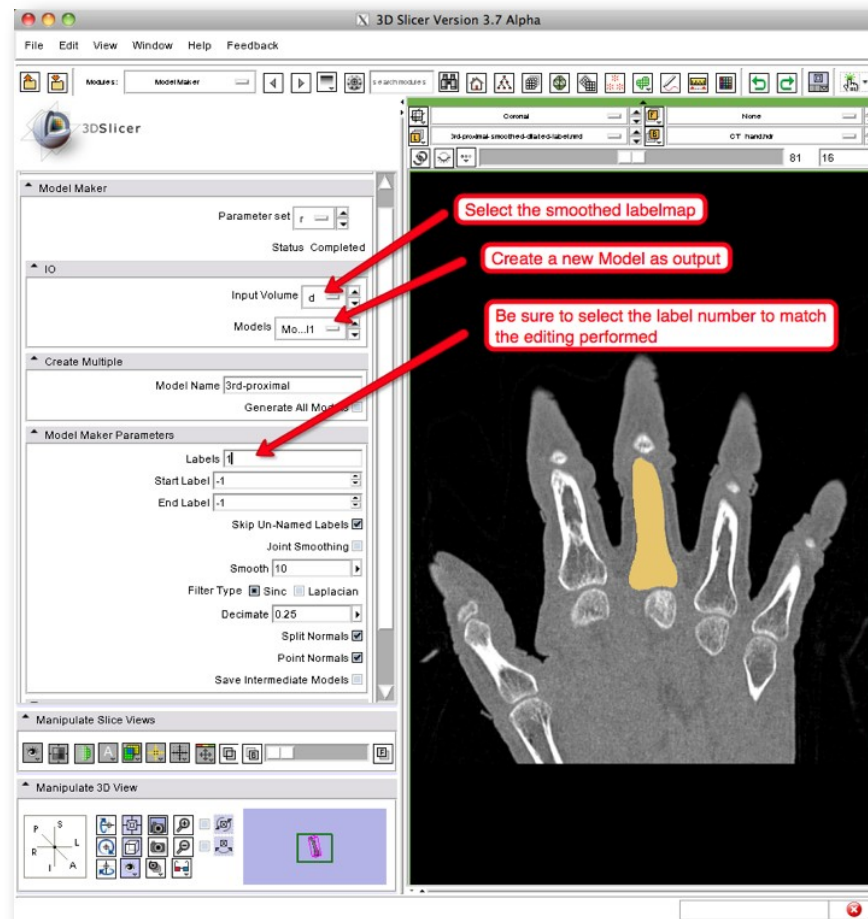
- After editing is complete, smooth the label map to reduce jagged edges on the voxel contour. Use the Label Map Smoothing module
- A smooth label boundary generates the best quality final mesh





# Create a Boundary Surface

- Once label map editing is complete, a boundary detection algorithm is run to wrap a contour around the label map
- Select the ModelMaker module to create the surface

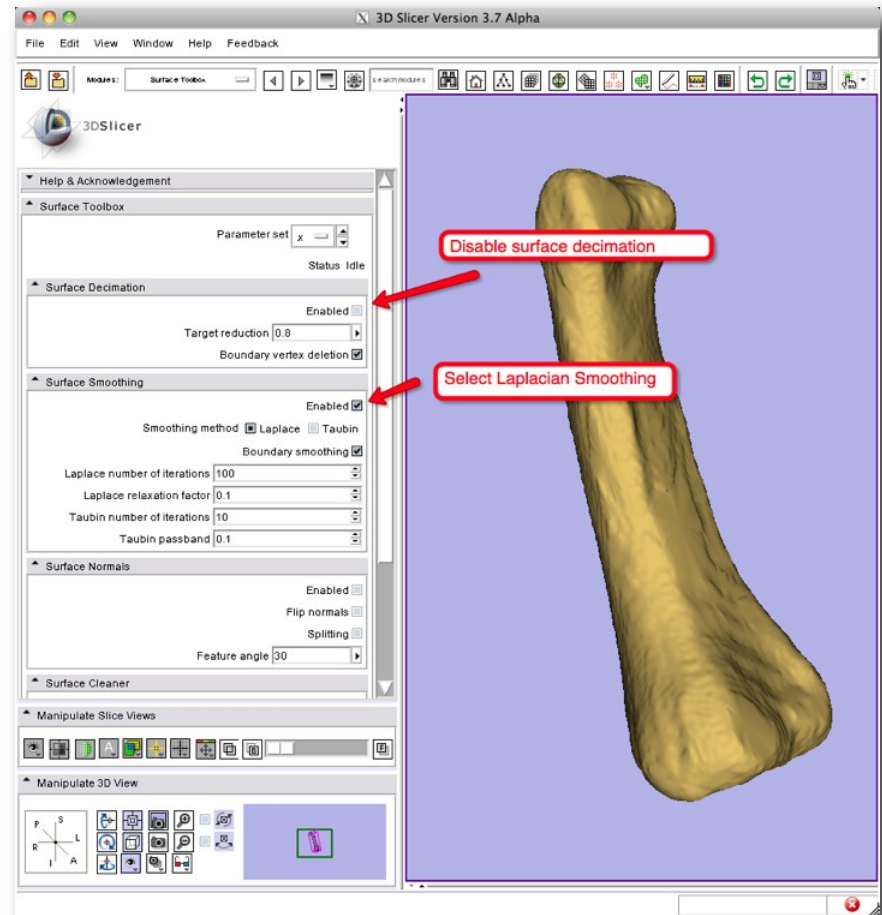






# Smooth Even More?

- The Surface Toolbox can smooth the polygonal surface of the model further if needed
- Remember, a smooth boundary makes for higher quality boundary elements in the volume mesh





# Loading a Surface

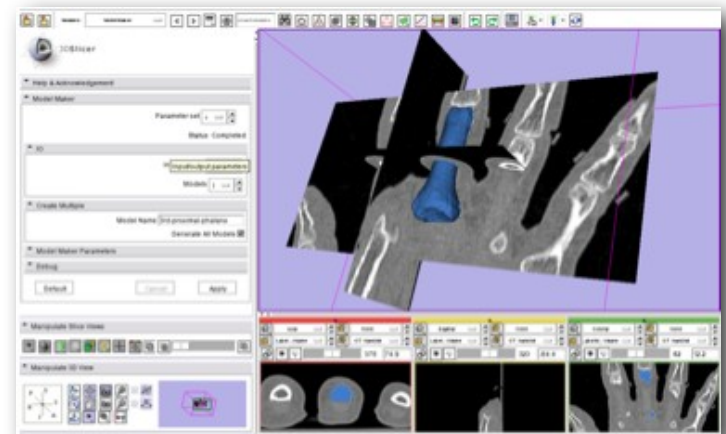
- The IA-FEMesh specific part of the workflow can begin with a surface loaded from an external file OR
- a Slicer3 Model is imported from the current Slicer3 session

STL, VTK, or VTP file



Current Slicer3 Session (MRML Scene)

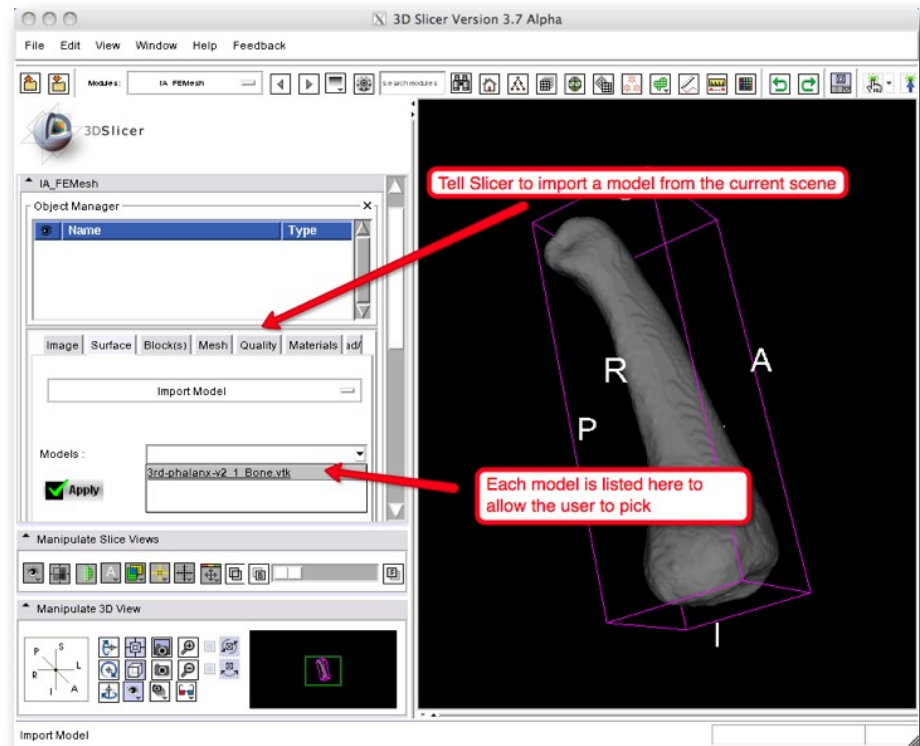
Import Model





# Importing a Surface Model

- Any Model object in the current Slicer MRML scene can be imported into the meshing module to start a meshing workflow
- Select the Import Model selection from the Surface tab in IA-FEMesh





# Creating a Building Block

---

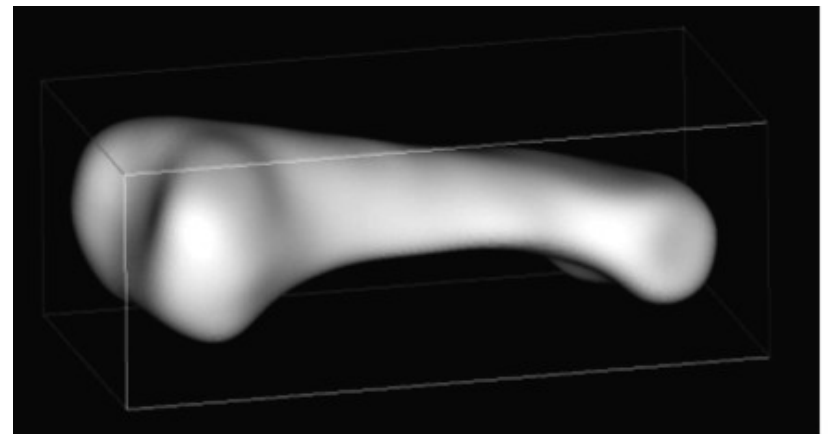
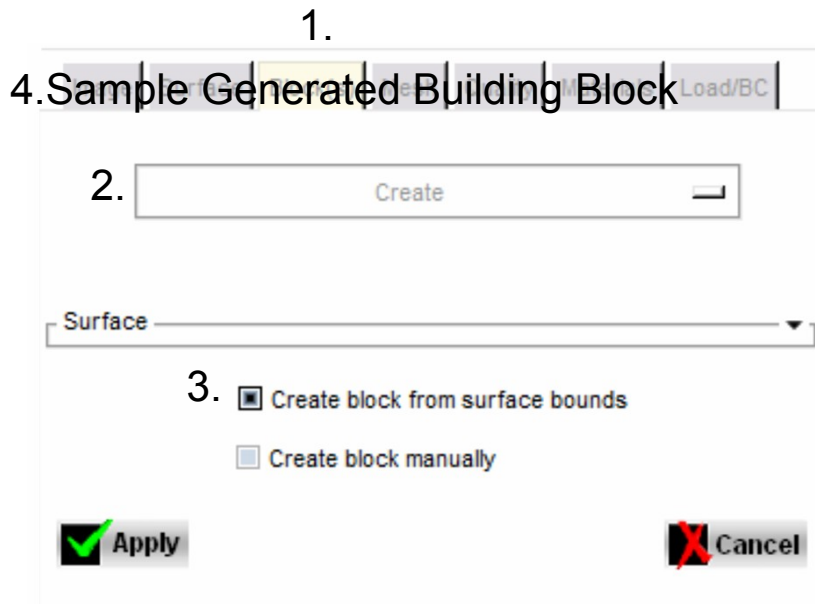
- The surface is surrounded by Building Blocks to represent the shape to the meshing algorithm
- Building blocks are made through an interactive widget editing process
- Please refer to the screencast as an example of the steps:

–<http://wiki.na-mic.org/Wiki/index.php/File:Meshing-2-building-blocks.mov>




# Creating a Building Block

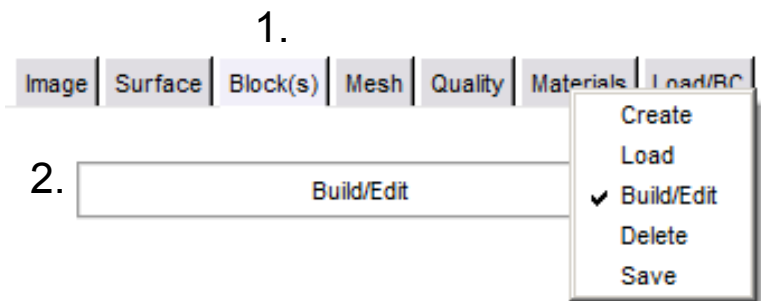
1. Select the Block(s) Tab
2. Select Create from the drop down menu
3. Check Create Block from Surface Bounds
4. Click Apply





# 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



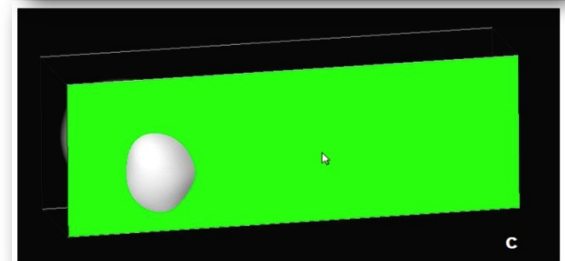
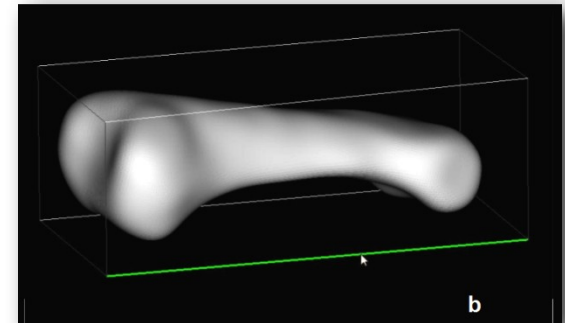
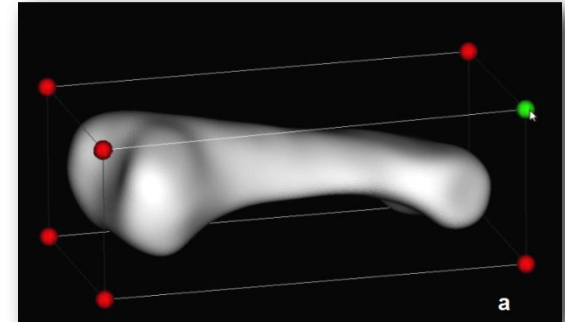
3a.



3b. Scale Spheres:



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





# Creating a Mesh

---

- The surface imported into IA-FEMesh along with the building block surrounding the surface are used to create the mesh
- Mesh elements are created by projecting the bounding box mesh seeds through the volume
- Please refer to the screencast as an example of the process:

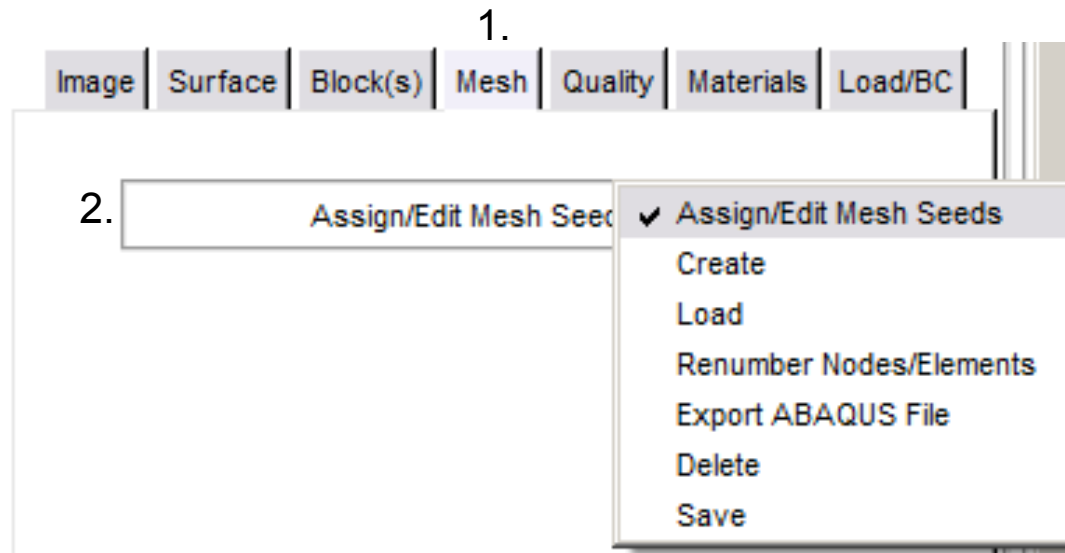
–<http://wiki.na-mic.org/Wiki/index.php/File:Meshing-3-mesh-creation.mov>






# Assigning Mesh Seeds

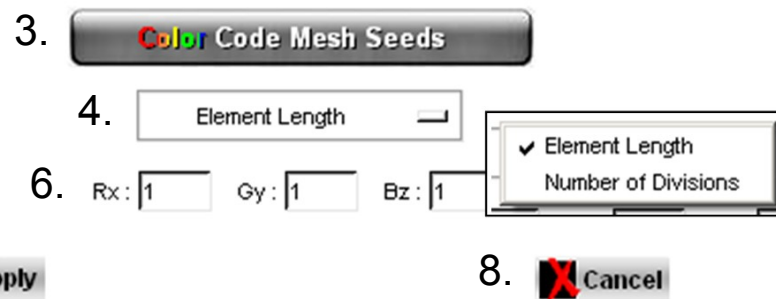
1. Select the Mesh tab
2. Select Assign/Edit Mesh Seeds from the drop down menu



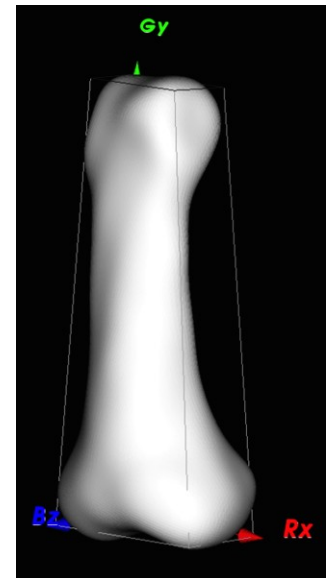


# Assigning Mesh Seeds (2)

3. Select the  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




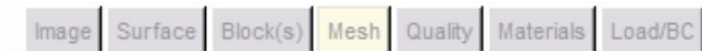
5.











# 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

1. 


2. 


3. ☒ Volumetric Mesh    
☐ Surface Mesh  

4. 

Surface & Building Block 

	#	Label
5. Node	1	
Element	1	

6. Interpolation: 

7. Perform Smoothing ☐ 



# Check Mesh Quality

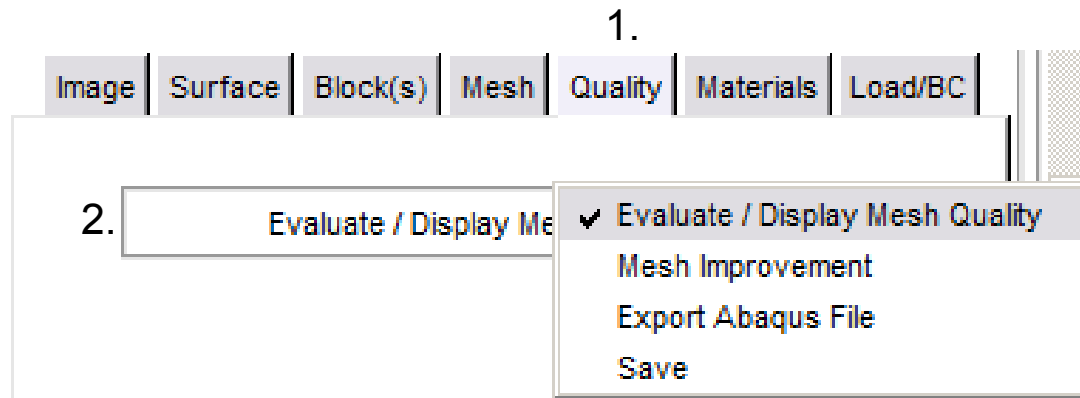
---

- Once mesh creation is complete, advance to the Quality tab on the interface to review the quality according to several metrics
- The process is demonstrated in the following screencast
  - <http://wiki.na-mic.org/Wiki/index.php/File:Meshing-4-mesh-quality-tab.mov>



# Checking Mesh Quality

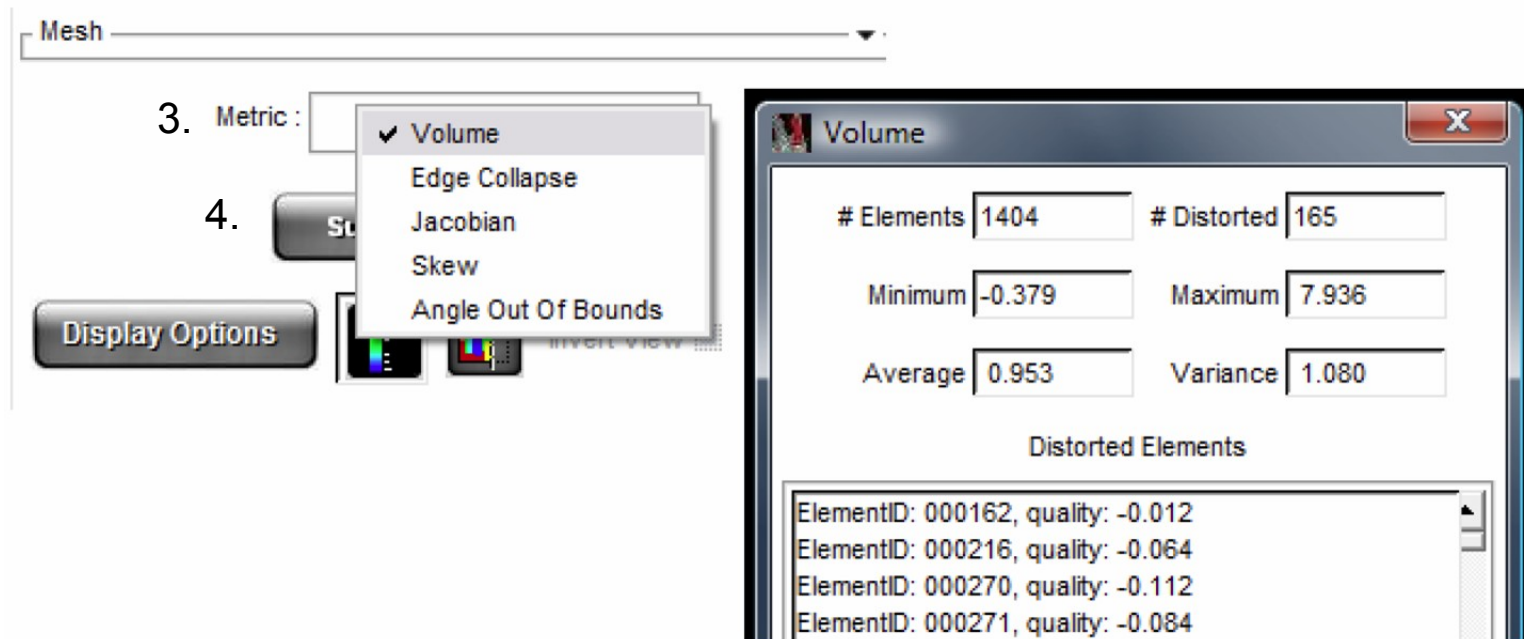
1. Select the Quality tab
2. Select Evaluate/Display Mesh Quality from the drop down menu






# Checking Mesh Quality (2)

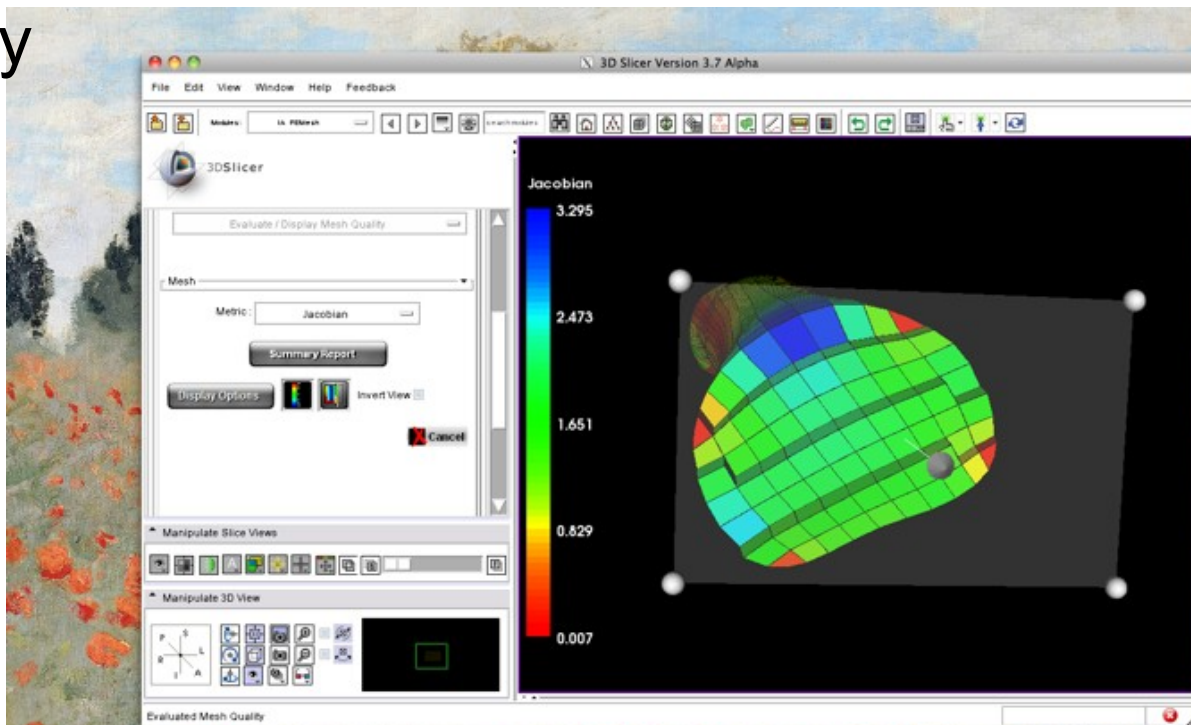
3. Select your Metric of Choice from the Metric drop down menu
4. Click the **Summary Report** button
5. Close summary report window after viewing
6. Click Cancel to exit the mesh quality check module





# Check Interior Elements

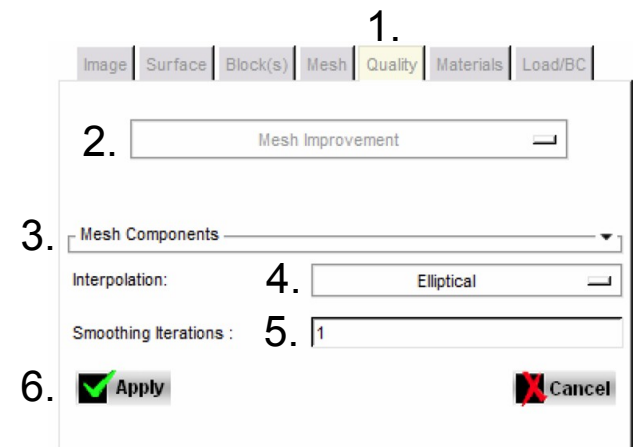
- Use the Cut Plane button  to examine interior elements by dragging the plane interactively





# 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



# After Mesh Generation


---

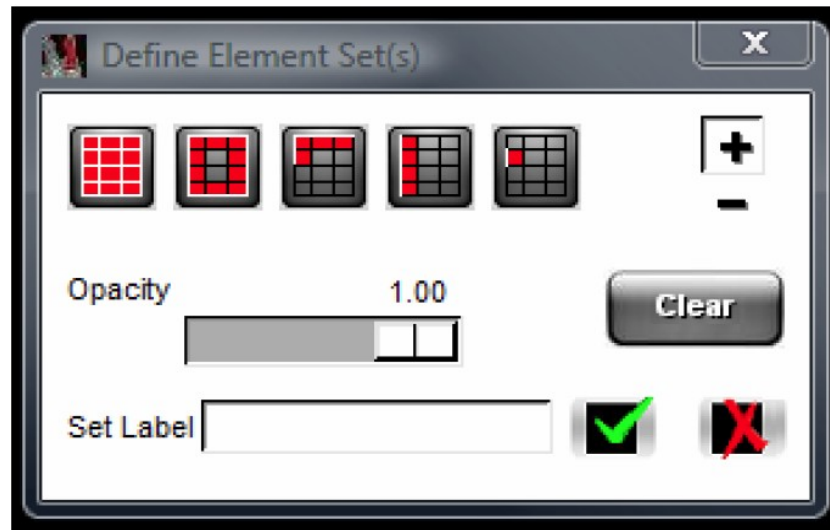
- After mesh generation is complete, the user can add material properties and loading conditions to prepare for external finite element solver execution
- The following slides cover these options. For additional detail, refer to the IA-FEMesh Users Guide

–[https://mri.radiology.uiowa.edu/downloads/IA-FEMesh\\_Manual\\_version1.pdf](https://mri.radiology.uiowa.edu/downloads/IA-FEMesh_Manual_version1.pdf)



# Assigning Material Properties





1. Select the Materials tab
2. Select the User-Defined option from the main menu
3. To add additional element sets, press 
  - A number of options are available for selecting element sets

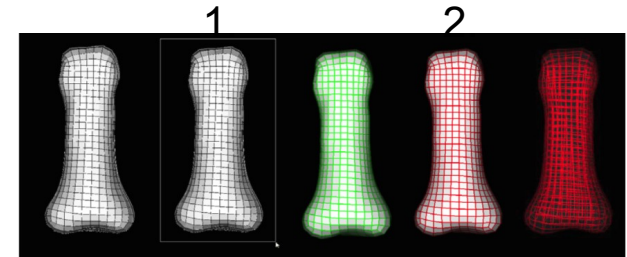




# Material Properties (2)

## 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





# Assigning Material Properties



- Material Property Assignments
  1. Select an element set using the “Element Set” drop down menu
  2. Enter the desired Modulus
  3. Enter the desired Poisson’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 :  1.


Modulus :  2.

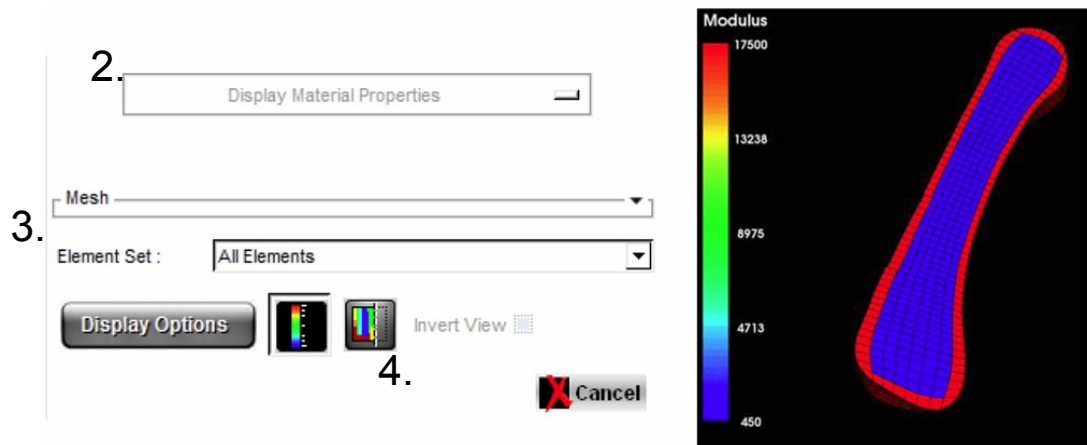
Poisson's Ratio :  3.

 **Apply** 4.  **Cancel** 6.





# Assigning Material Properties

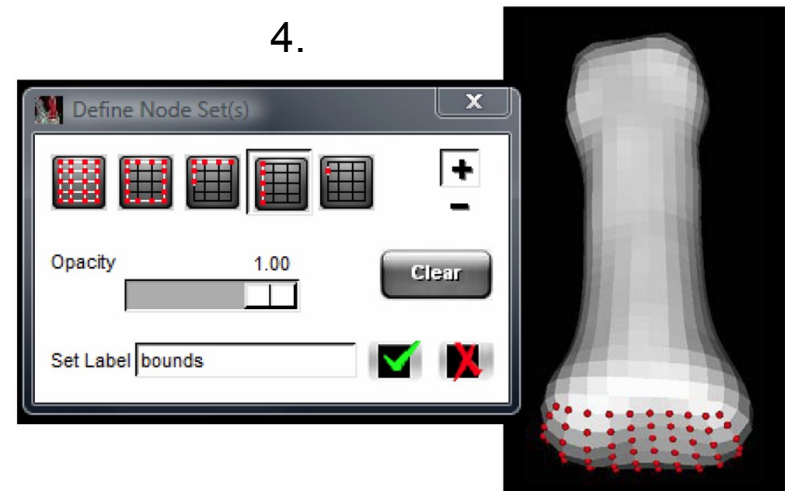
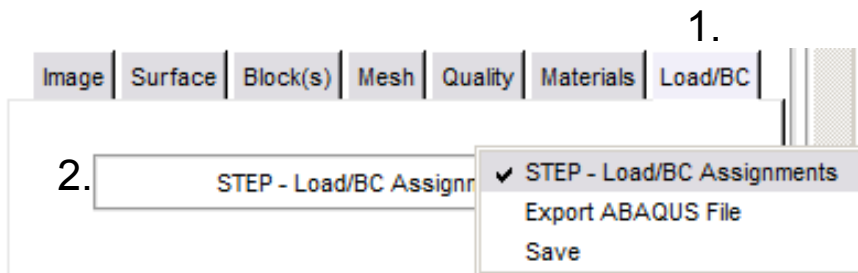
- Visualizing the Material Property Assignments
  1. Select the Materials tab
  2. Select Display Material Properties from the drop down menu
  3. Select the desired element set from the Element Set drop down menu
  4. Use the  button to display a cutting plane which may be manipulated to view internal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







# Load and Boundary Conditions

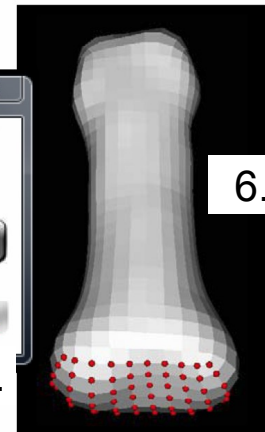
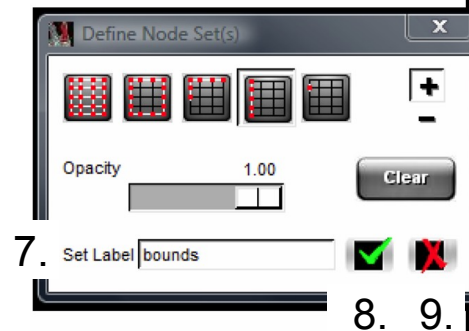
1. Select the Load/BC tab
2. Select the STEP – Load/BC Assignments from the drop down menu
3. Select 
4. Select  from the Node Set toolbar
  - This enables the nodes associated with a face of a building block to be readily chosen





## 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 







# Load and Boundary Conditions (3 )

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

Node Set : bound

X:  Y:  Z:

Force  
✓ Displacement  
Rotation  
Moment

11.

Displacement

12. Node Set : bounds

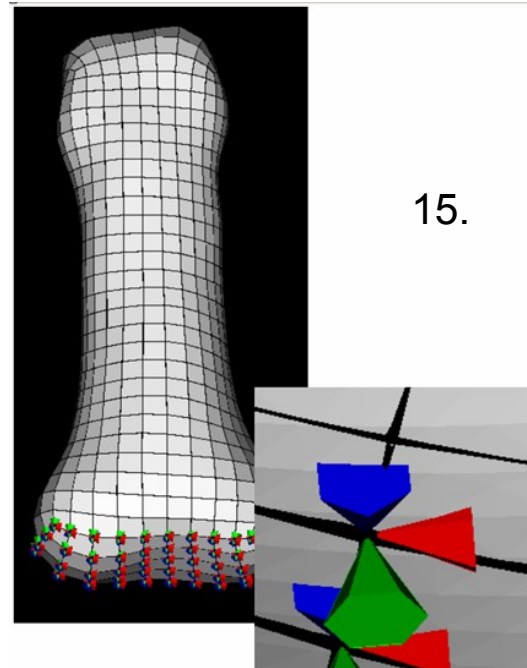
13. X:  Y:  Z:



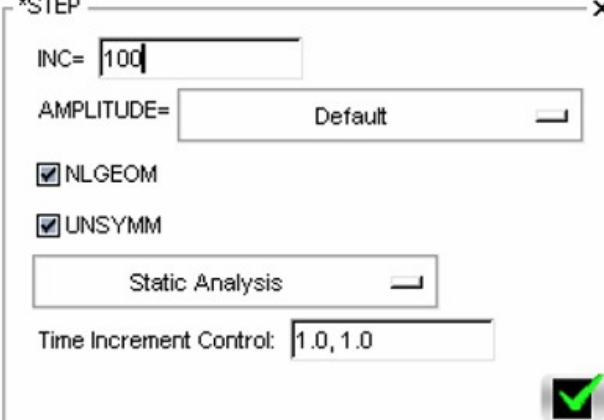
## Load and Boundary Conditions (4)

---

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



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



\*STEP

INC= 100

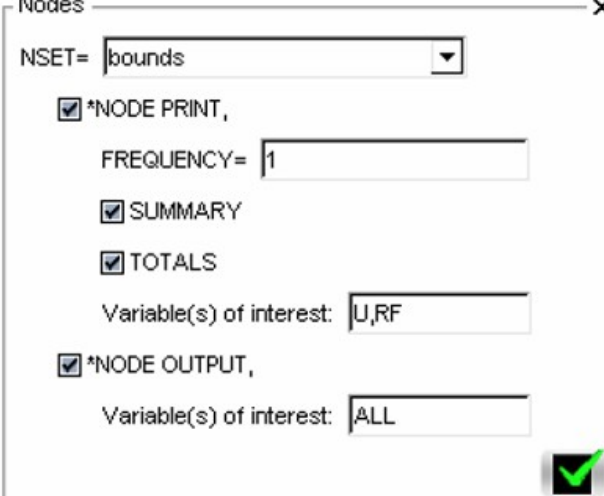
AMPLITUDE= Default

☒ NLGEOM

☒ UNSYMM

Static Analysis

Time Increment Control: 1.0, 1.0



Nodes

NSET= bounds

☒ \*NODE PRINT,  
FREQUENCY= 1

☒ SUMMARY

☒ TOTALS  
Variable(s) of interest: U,RF

☒ \*NODE OUTPUT,  
Variable(s) of interest: ALL

- 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



# Review of Workflow Steps

---

- Input an image
- Create, smooth, (and optionally dilate) the labelmap
- Use ModelMaker to create a polygonal surface model
- Use Surface Tools to further smooth model polys using Laplacian smoothing
- Import the model into IA-FEMesh using Surface/Import Model from the MRML scene
- Create Building Blocks surrounding the surface
- Assign mesh seeds and Create the Mesh
- View the mesh quality
- Improve the mesh quality through smoothing
- Add Material Properties
- Add Initial Boundary Conditions
- Export to FE Solver (Abaqus)



# Conclusion

---

- It is now possible to perform end to end processing from source imagery all the way through Finite Element mesh generation, mesh analysis, and mesh optimization using Slicer3
- Volumetric meshes for objects can be created through a simple, intuitive process
- Any Slicer3 model can be imported to start the mesh generation process



# Acknowledgments

---



National Alliance for Medical Image Computing  
NIH U54EB005149



National Institute of Biomedical Imaging and Bioengineering  
R21EB001501 and R01EB005973



Musculoskeletal Imaging Modeling and EXperimentation (MIMX)  
Laboratory  
The University of Iowa