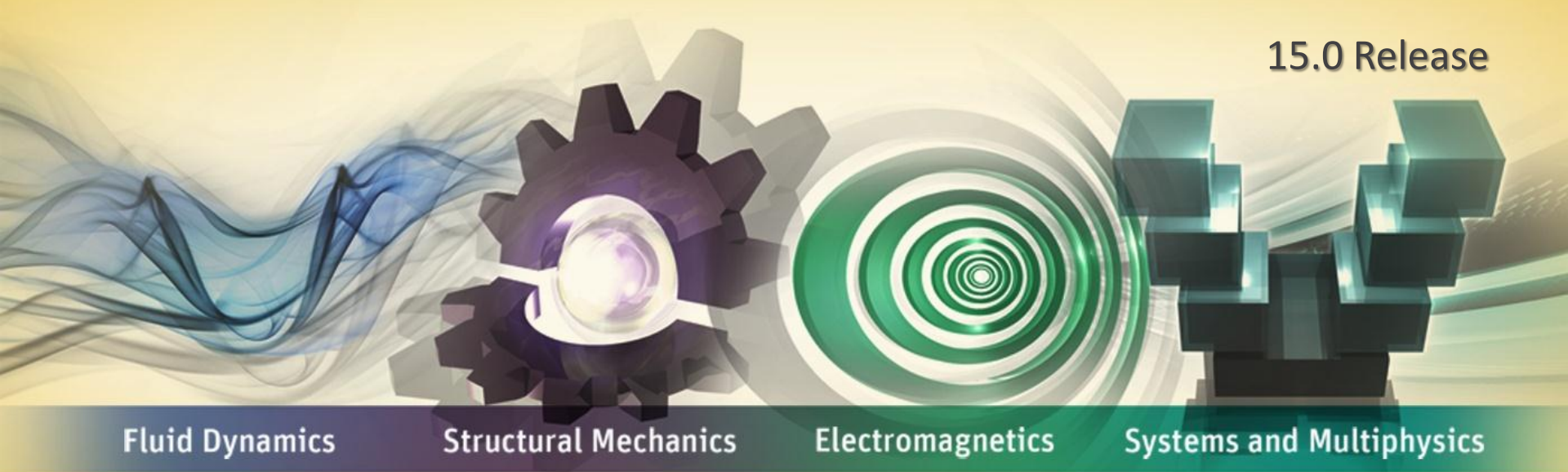


Workshop 4.3

Body Fitted Cartesian - Femur

15.0 Release

A horizontal banner with a yellow-to-orange gradient background. It features four distinct 3D visualizations: blue fluid flow lines on the left, a purple gear with a glowing center, a green concentric circle pattern, and a stack of teal cubes on the right. Below these images is a dark teal bar containing four white text labels.

Fluid Dynamics

Structural Mechanics

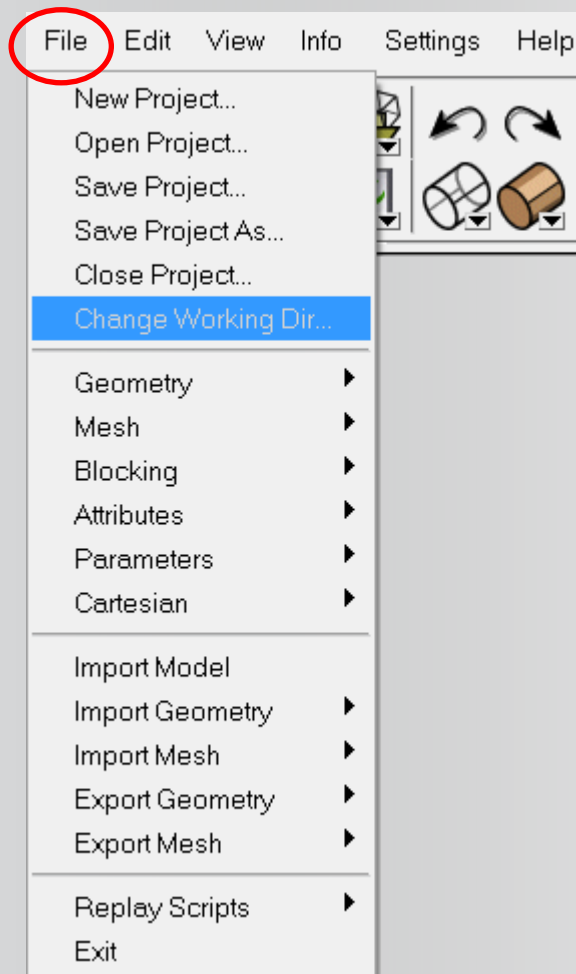
Electromagnetics

Systems and Multiphysics

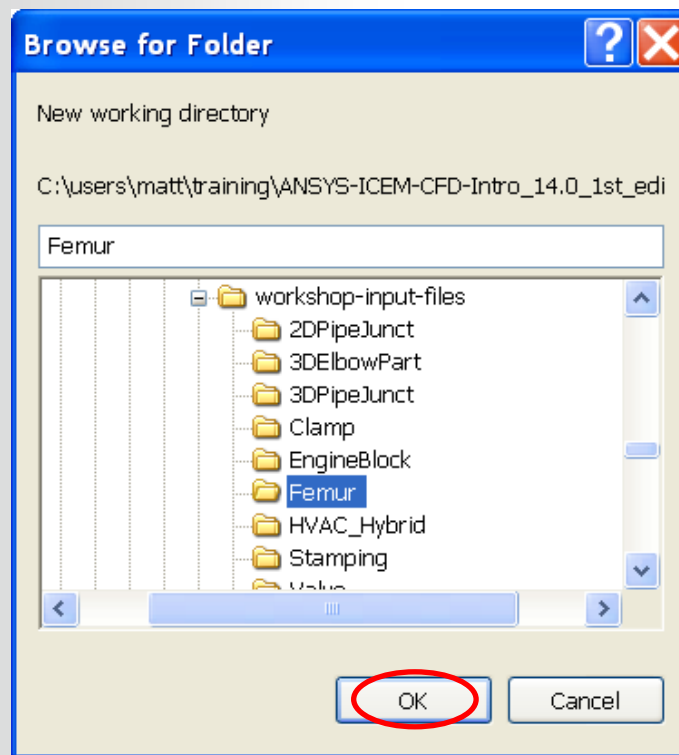
Introduction to ANSYS

ICEM CFD

Set Working Directory



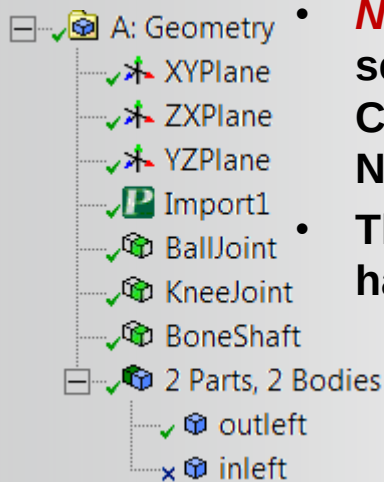
- Set working directory
 - **File > Change Working Dir...**
 - Choose **Femur**
 - Click **OK**



Import Geometry

- **File -> Import Model**

- Select the file **LeftFemur.agdb** (Design Modeler Geometry)
- Click the **“Open”** button
- Before importing the Geometry, the user is presented with options
- Select on **Create Subset(s) from Named Selections**

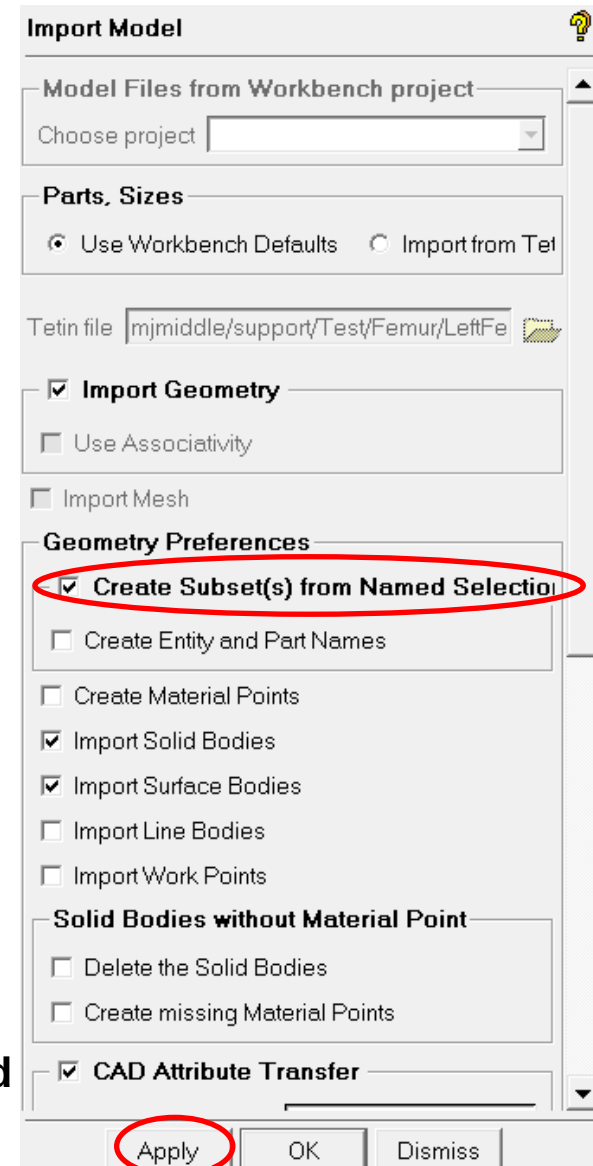


- **Named Selections** can be setup in DM, Simulation or CAD packages such as UG NX, ProE, or SolidWorks

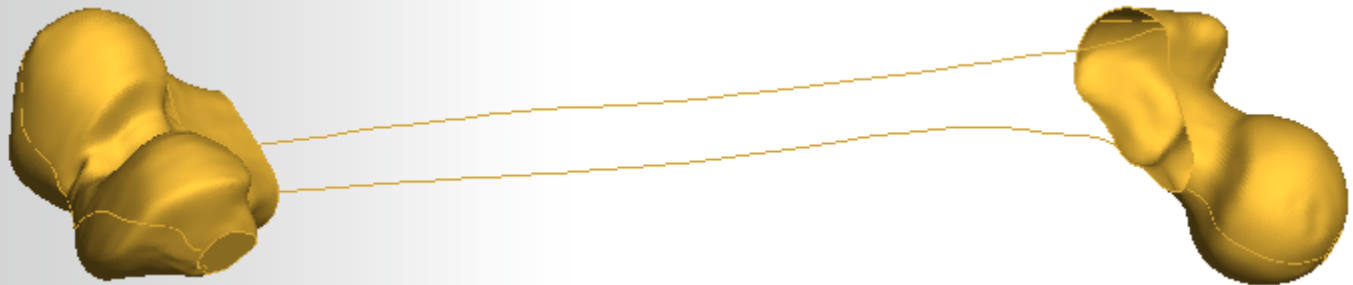
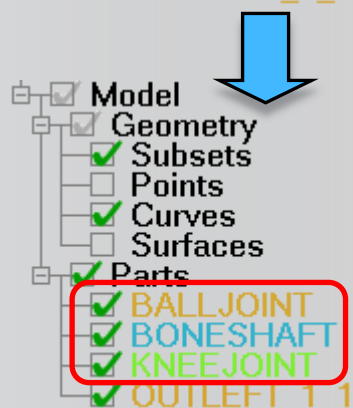
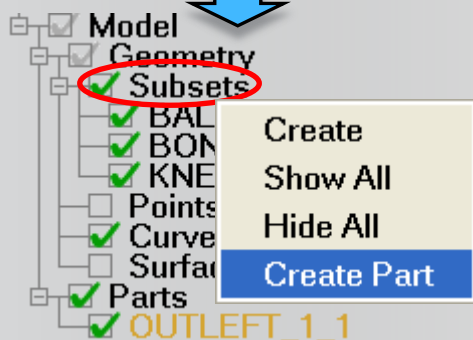
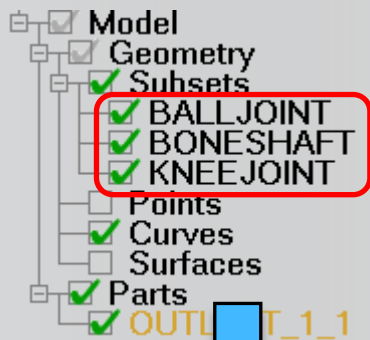
- Three **Named Selections** have been defined in DM

- Balljoint
- KneeJoint
- BoneShaft

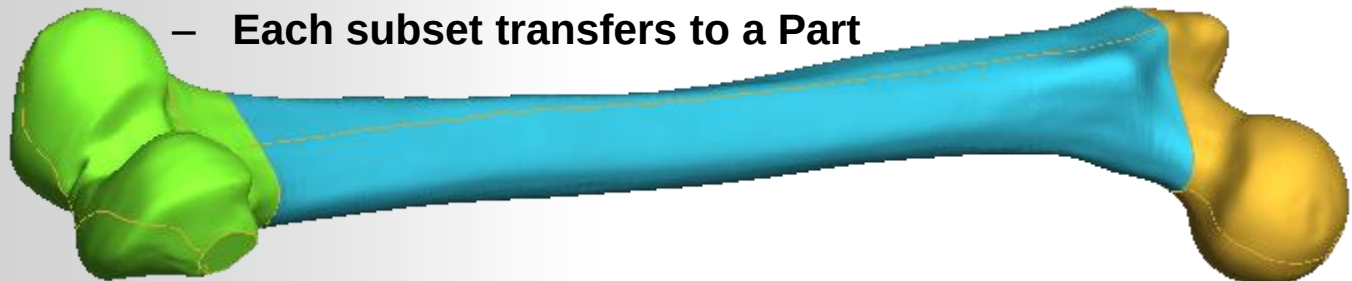
- Accept all the other defaults and click **Apply**



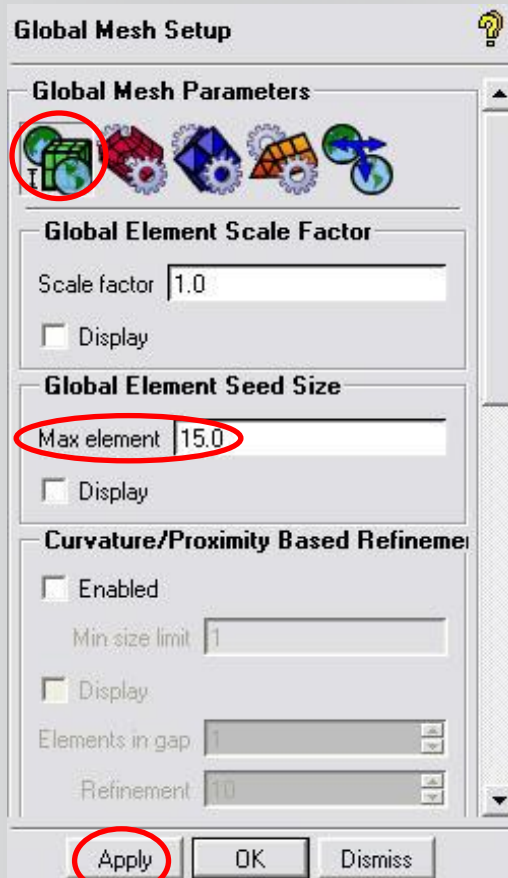
Create Parts from Subsets



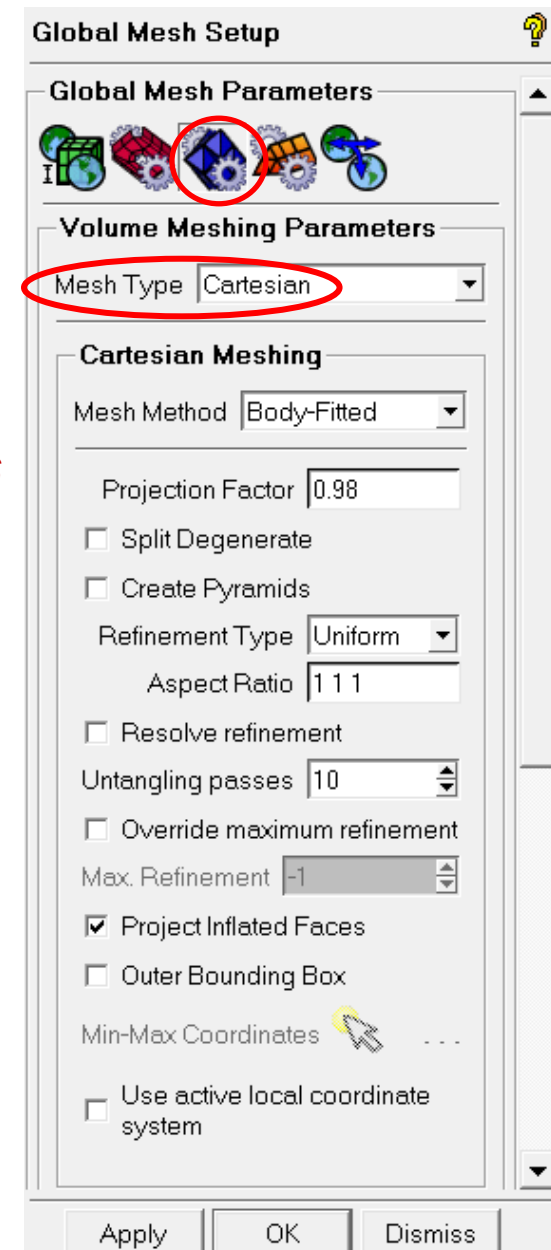
- The **Named Selections** appear as geometry **Subsets**
 - These can be used to control display or selection
 - These can be turned into Parts
 - The same geometry entity can be in multiple subsets while it can only be in one part at a time, so only the first subset with a duplicate entity may be converted to a part
- Convert subsets to parts
 - Make sure all the Subsets are active
 - Right Click on **“Subsets”** and choose **“Create Part”**
 - Each subset transfers to a Part



Global Mesh Setup

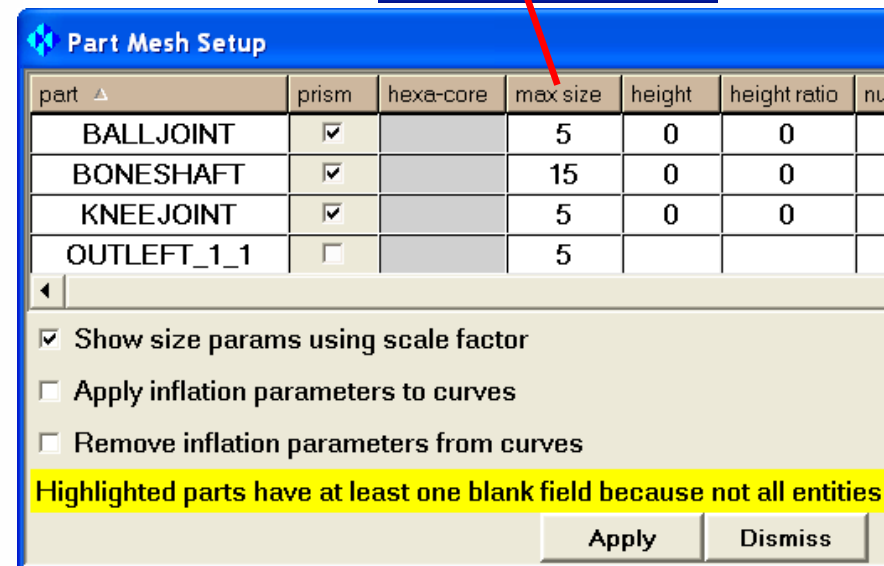
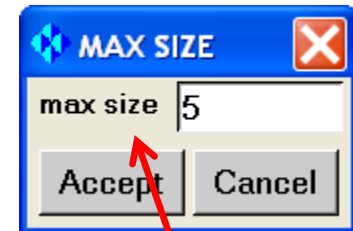


- **Mesh > Global Mesh Setup > Global Mesh Size**
 - Set **Max element** = 15
 - Click **Apply**
- **Mesh > Global Mesh Setup > Volume Meshing Parameters**
 - Set the Mesh Type to **“Cartesian”**
 - Set the Mesh Method to **“Body-Fitted”**
 - Keep the defaults as shown
 - Refinement Type is set to **“Uniform”**
 - **Project Inflated Faces** is Checked
 - Click **Apply**

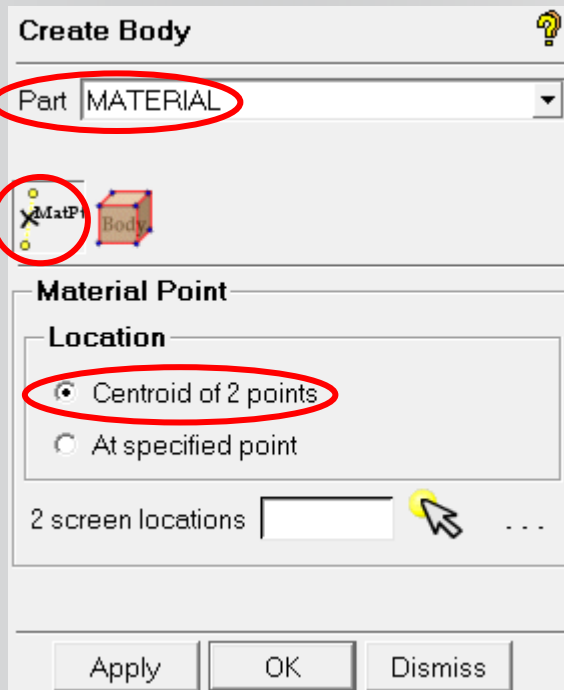


Part Mesh Setup

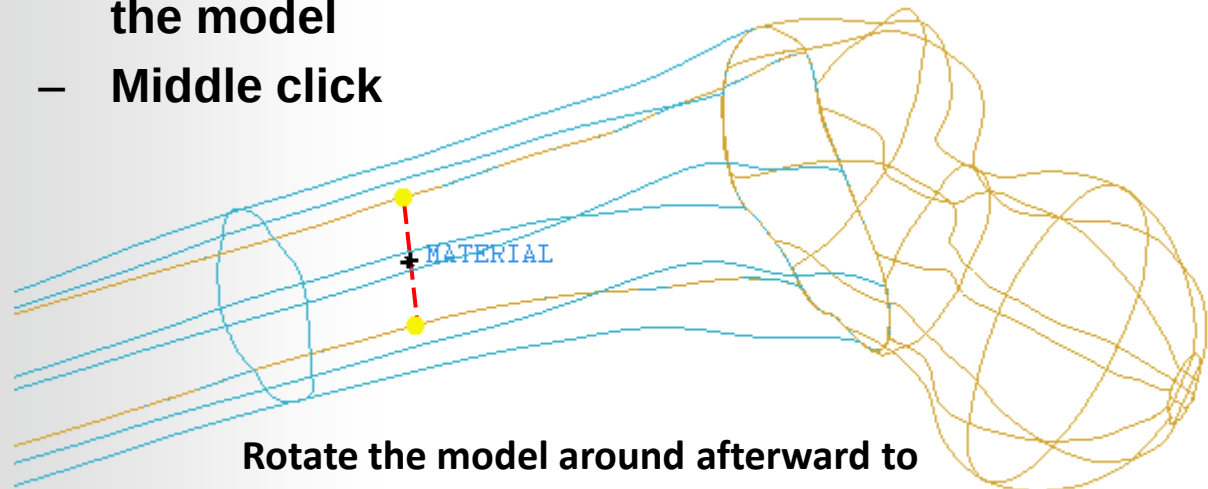
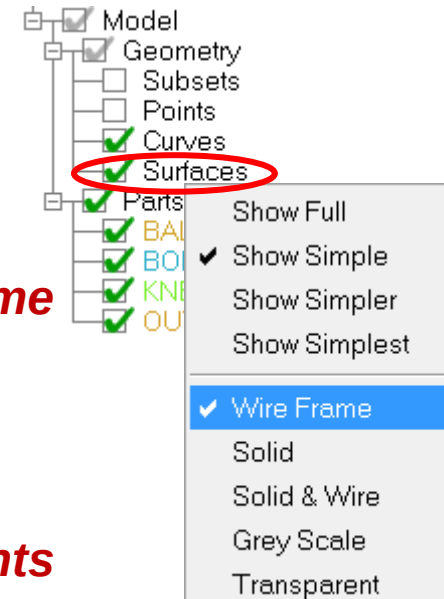
- Set mesh sizes and inflation
- **Mesh -> Part Mesh Setup**
 - This brings up the **Part Mesh Setup** dialog
- Left Click on the **maximum size** header
 - This brings up a dialog in order to set one value for the entire column
 - Set **max size** to **5**
 - Click **Accept**
- Left Click in the BONESHAFT **max size** field
 - Change the max size of **5** to **15**
- Left Click on the **Prism** header
 - This toggles prism on/off for all parts
- Deselect Prism option for OUTLET_1_1
- Click **Apply**
- Click **Dismiss**
- The prism control is used to control which surfaces are inflated in BFCart
 - Only one layer can be inflated



Create Material Point



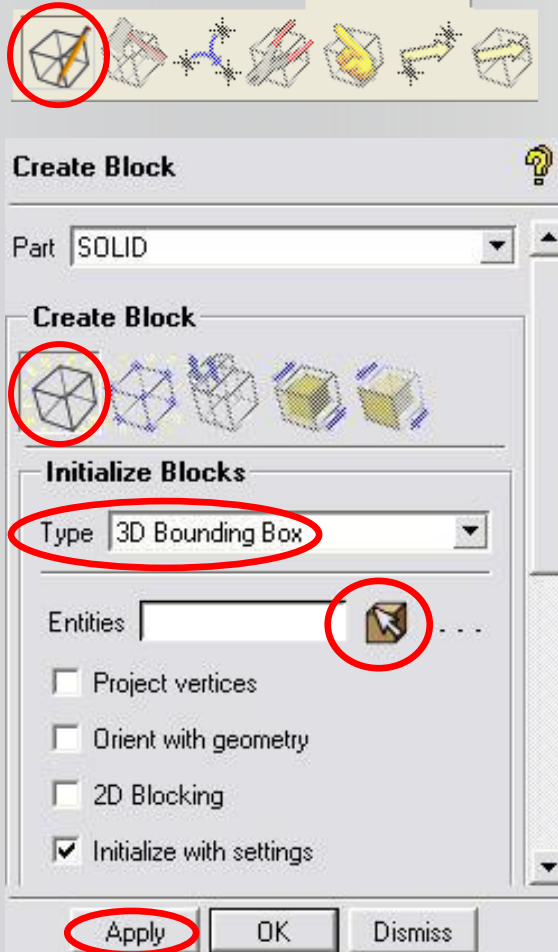
- Turn on **Surfaces**
- Right click on **Surfaces** in the Model tree and turn on **Wire Frame**
- **Geometry > Create Body > Material Point**
 - Enter MATERIAL for the **Part**
 - Location = **Centroid of 2 points**
 - Select 2 locations so that the center is inside the model
 - Middle click



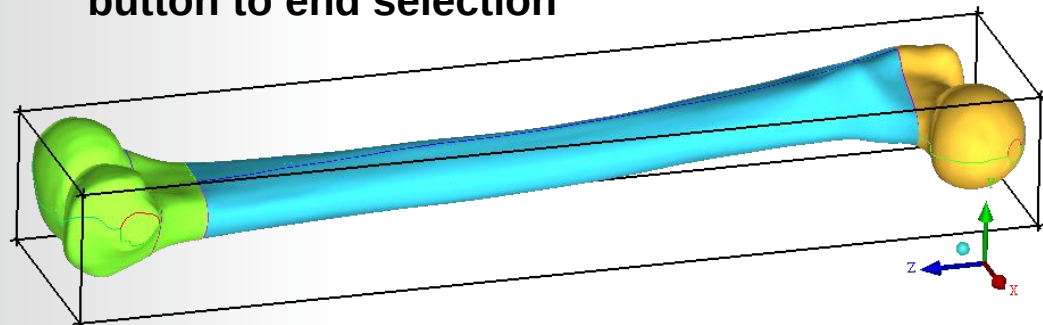
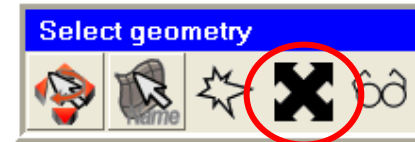
Rotate the model around afterward to make sure the material point is inside the model

Initialize hexa blocking

Blocking



- **Blocking -> Create Block**
 - This step is optional, BFCart could create its own cartesian input mesh
 - This enables us to control the biasing with ICEM CFD Hexa's advanced edge parameter and splitting functionality (discussed more in hexa section)
- **Initialize Blocks**
 - **Type** should be set to **3D Bounding Box**
 - We just want a rectangular box to represent cartesian space
 - Click the **"select geometry"** icon 
 - Click on the **"all"** icon in the Select geometry toolbox
 - Or use the **"a"** hotkey
 - Click the middle mouse button to end selection

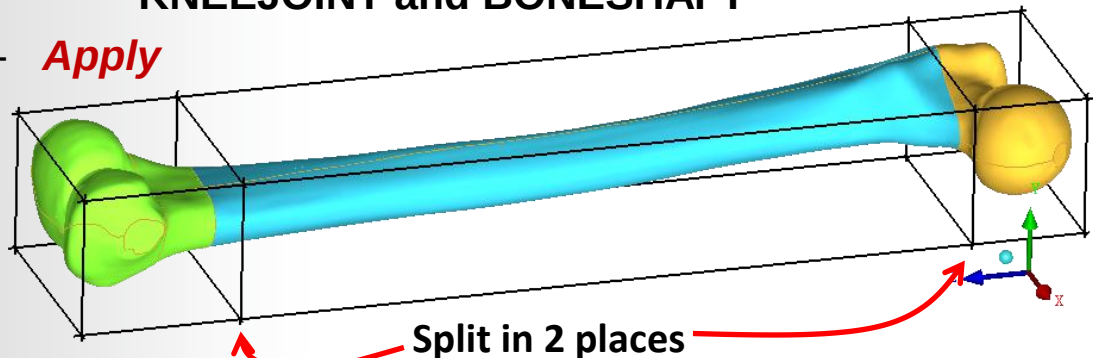
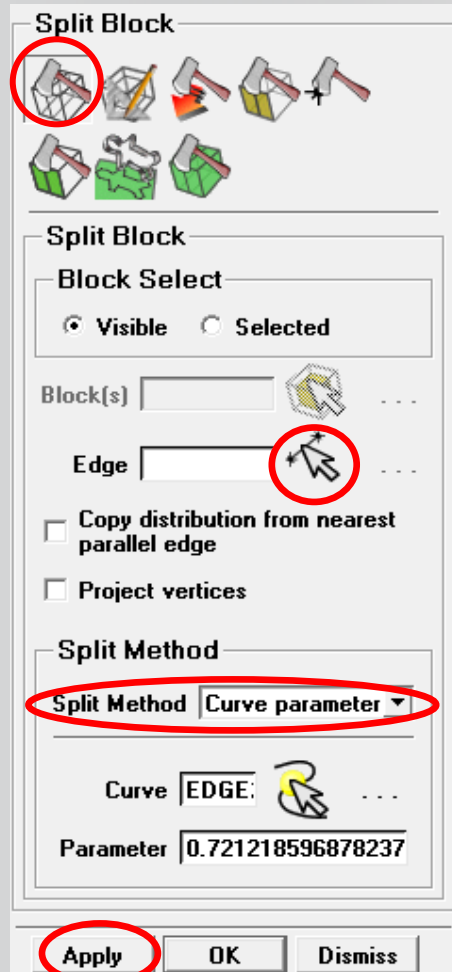


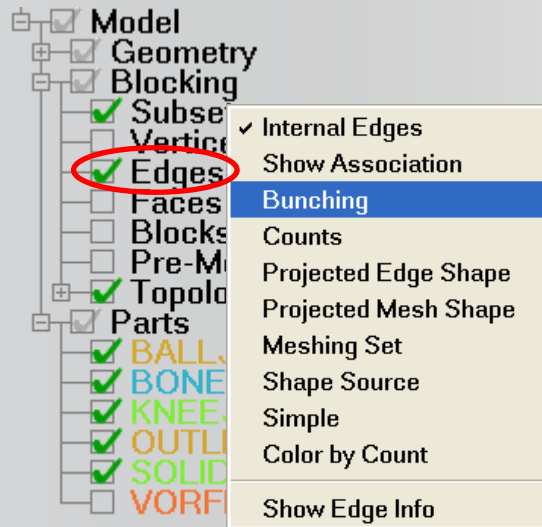
Split Block

Blocking

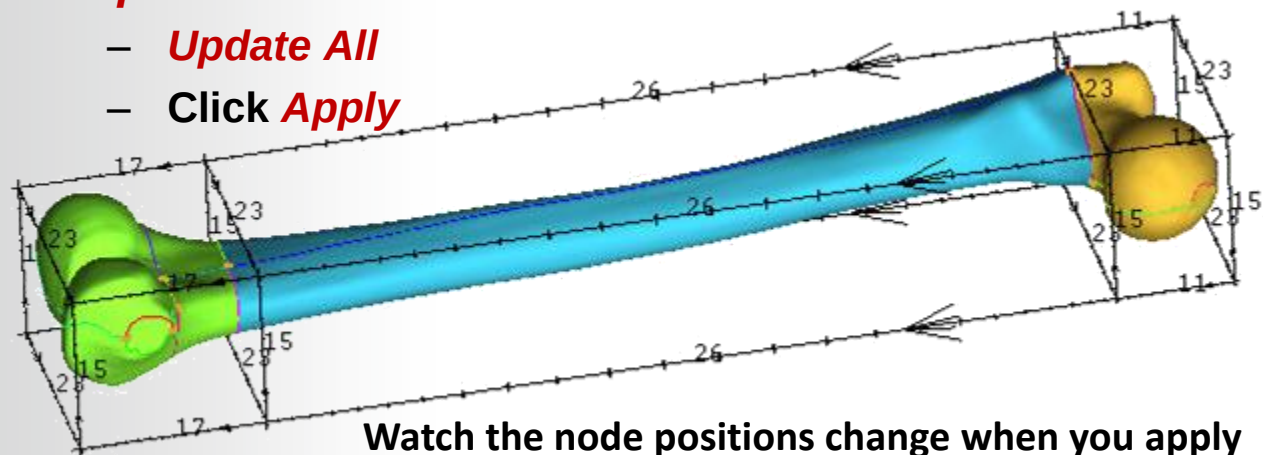
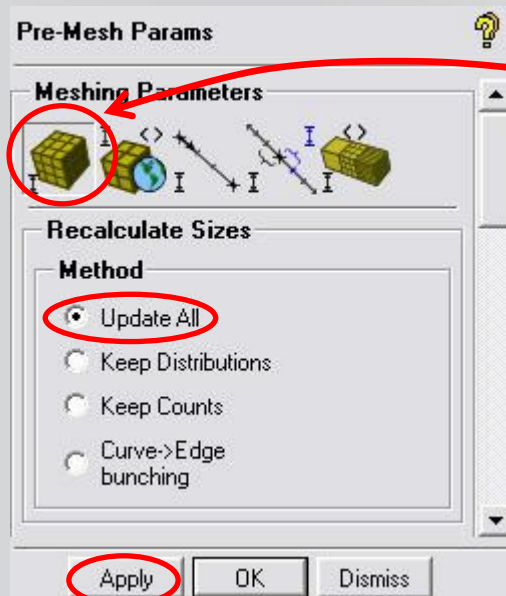


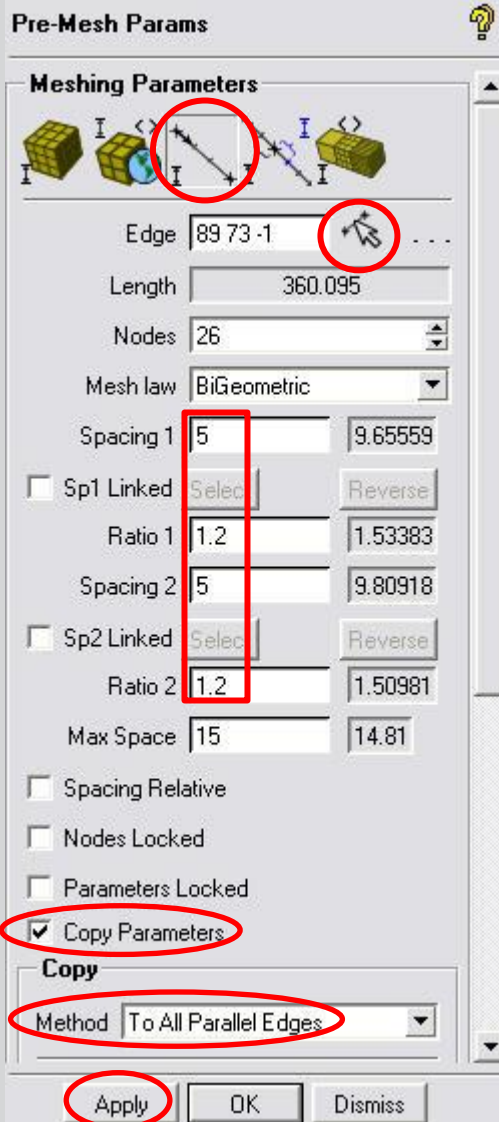
- The two primary ways to create Cartesian Stations are splitting and Edge Params
- **Blocking > Split Block**
 - Set the **Split Method** to **Curve parameter**
 - Click on the “**Select edges**” icon 
 - Left click on one of the 4 longest edges
 - It will now switch to selecting the curve, so select any location along the curve dividing the BALLJOINT and BONESHAFT
 - **Apply**
 - Click on the “**Select edges**” icon again 
 - Left click one of the 4 longest edges again
 - Select anywhere along the curve between the KNEEJOINT and BONESHAFT
 - **Apply**



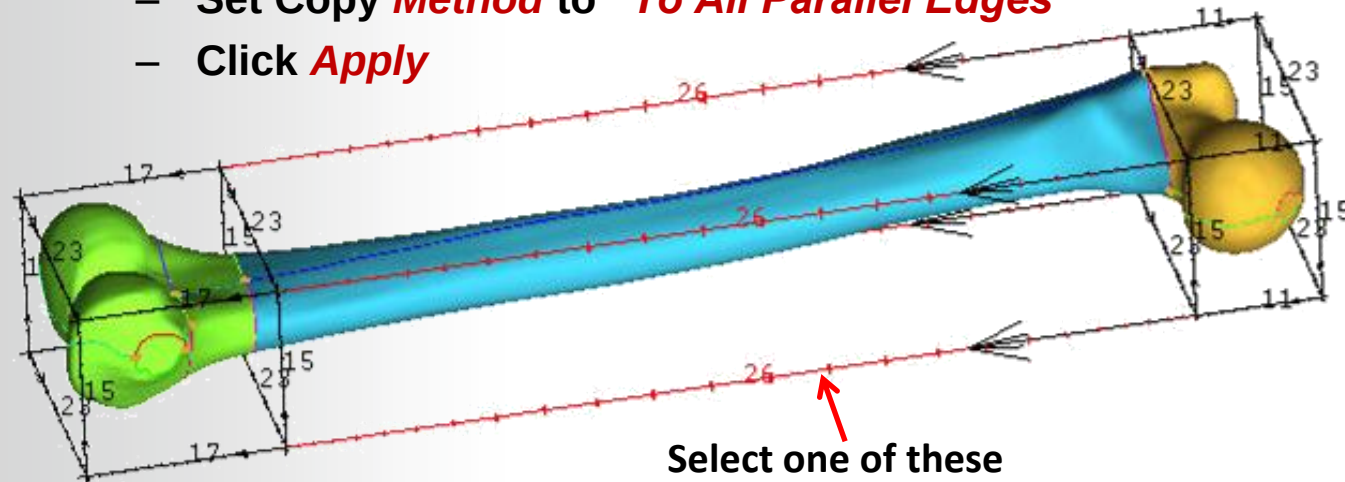


- Expand the **Blocking** Branch of the Model Tree
- Right click on **Edges**
 - Left click on **Bunching**
 - This will display Edge Bunching (Station Distribution)
- **Blocking -> Pre-Mesh Params**
 - The blocking is a separate entity from the geometry
 - This next step will transfer the sizes set on geometry (through **Part Mesh Setup**) to the blocking
- **Update Sizes**
 - **Update All**
 - Click **Apply**

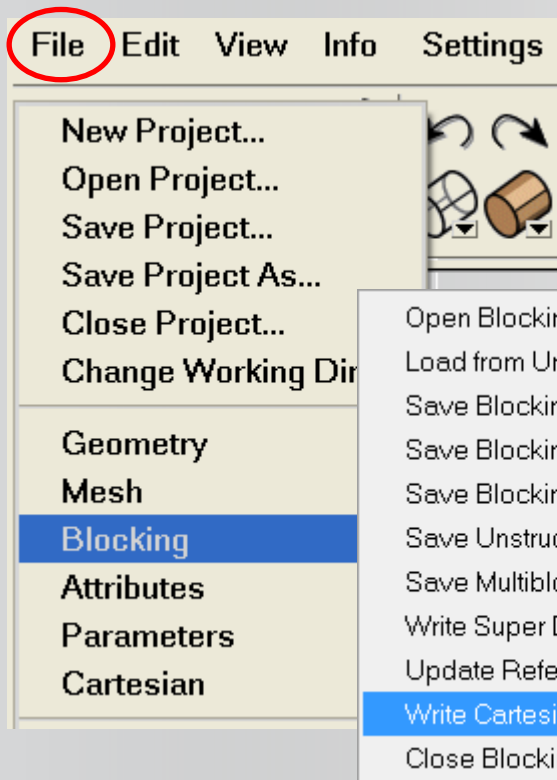




- **Blocking -> Pre-Mesh Params -> Edge Params**
- Click the “**Select edges**” icon and **Left Click** on one of the 4 longest edges (along the shaft)
 - Depending on your exact split locations, this should show approximately 26 Nodes
 - Change the **Spacing 1** and **Spacing 2** to “**5**”
 - This will match the spacing at the ends
 - Change the **Ratio 1** and **Ratio 2** to “**1.2**” (growth toward center)
 - Turn on **Copy Parameters**
 - Set Copy **Method** to “**To All Parallel Edges**”
 - Click **Apply**



Write Cartesian Grid



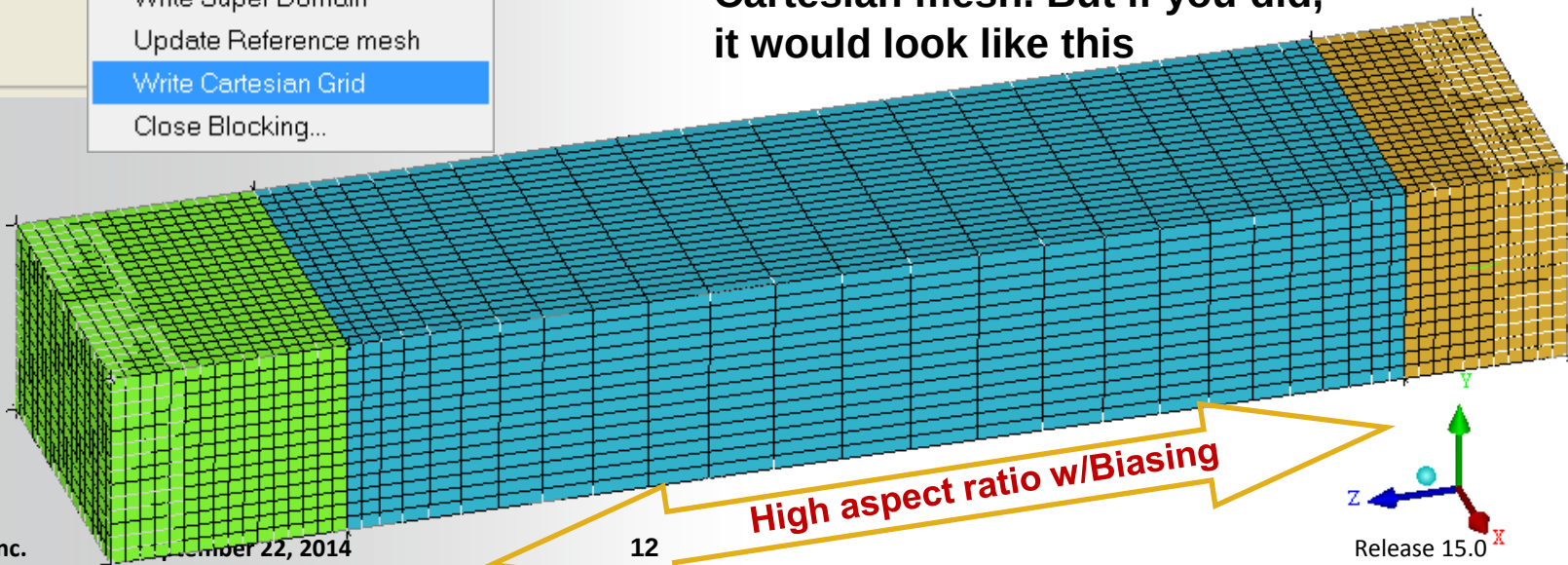
- **File -> Save project As**
 - Save as **Femur.prj**
- **File -> Blocking -> Write Cartesian Grid**
 - This converts the splits and biasing to Cartesian Stations

The file (**Femur.crt**) is saved in the project directory

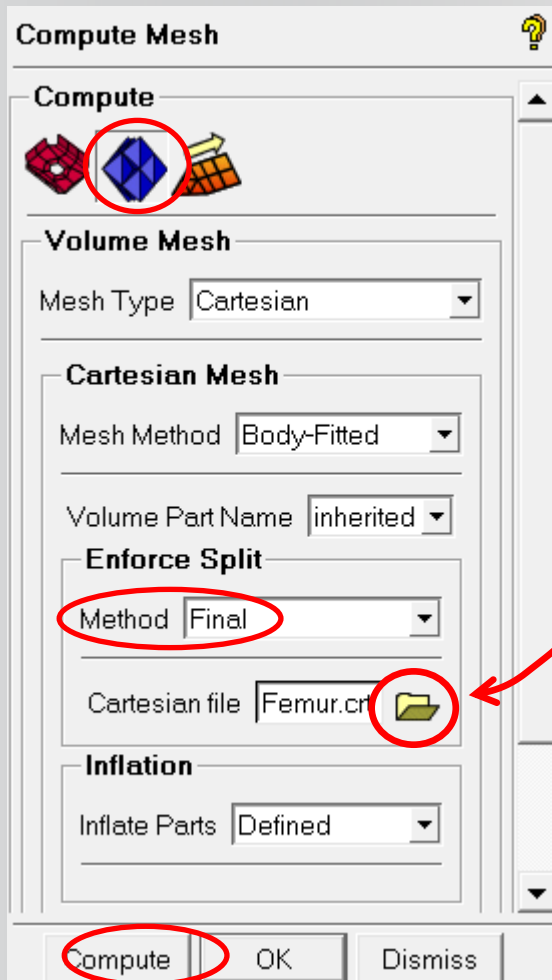
- Called "**blocking.crt**" if no project is opened or saved

You don't need to view the Cartesian mesh. But if you did, it would look like this

Pre-Mesh
computed
with **No**
Projection

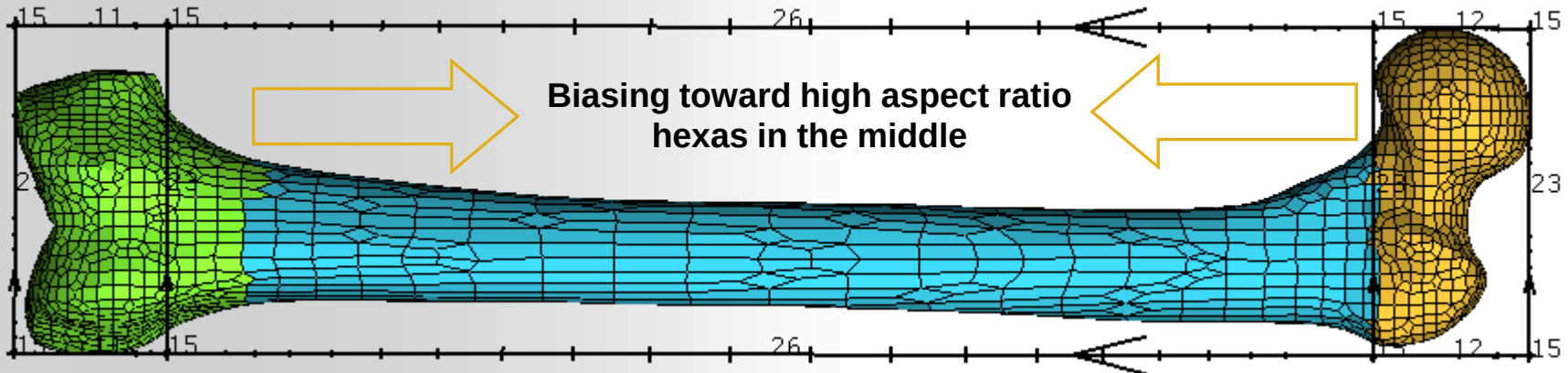


Compute Mesh

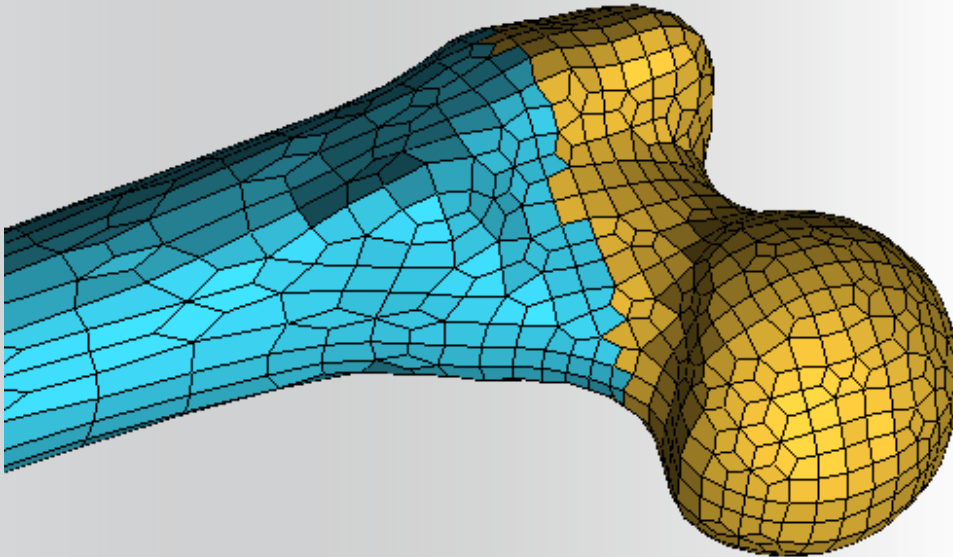


- Select **Mesh > Compute Mesh > Volume Mesh**
 - Mesh Type is automatically set to **“Cartesian”** and Mesh Method to **“Body-Fitted”** from global settings
 - Leave the **Volume Part Name** as **Inherited**
 - **Inherited** will use material point names
 - Set **Enforce Split** method as **“Final”**
 - This is why we made the cartesian file
 - Click on the File Browse icon and select the **Femur.crt** file we just created
 - **Note: one was also included in the tutorial files (FEMUR_Cart.crt)**
 - Ensure **Inflate Parts** is set to **“Defined”**
 - This will use the **Part Mesh Setup** parameters to control the inflation (prism column)
 - **Compute**
 - If you are prompted to save the tetin file, type a name and accept it
 - **BFCart** is a batch process that runs off the last saved geometry file

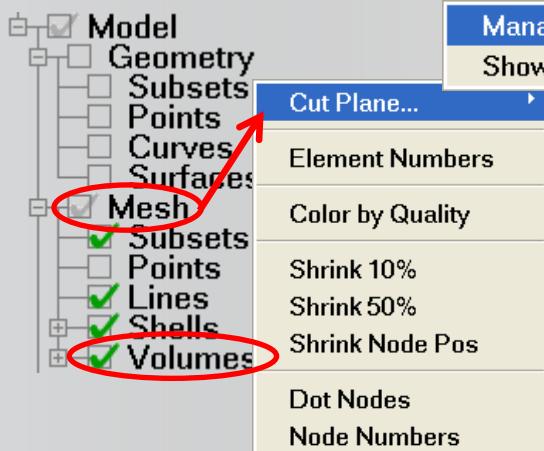
- The mesh should take about 30 seconds to generate.



- Note: this mesh is pure Hexa. There is a global option for creating pyramids which is useful if the model has sharp features inducing poor quality.
- Body Fitted Cartesian is best with “biomedical models” which are all curvy or “boxy” models such as some circuit boards
 - Hard corners cause severe problems with BFCart if they are not aligned along an XYZ direction



Cutplane View

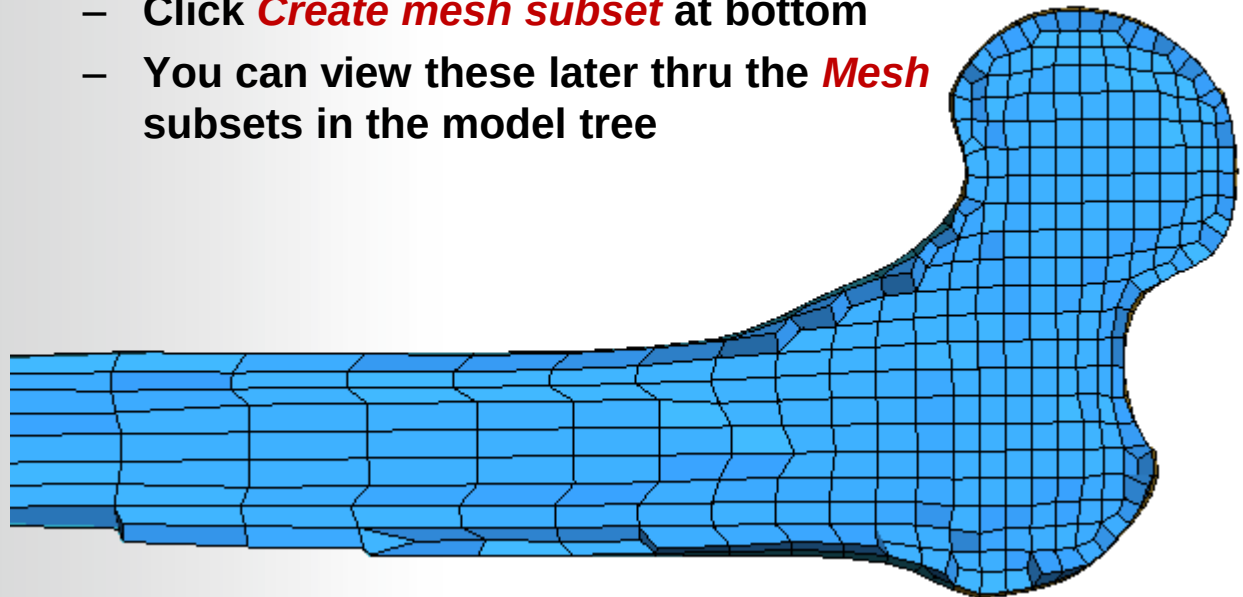
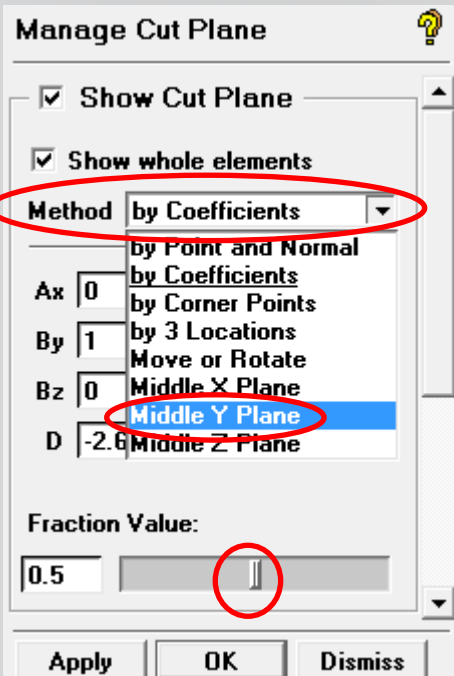


Manage Cut Plane

Show Cut Plane

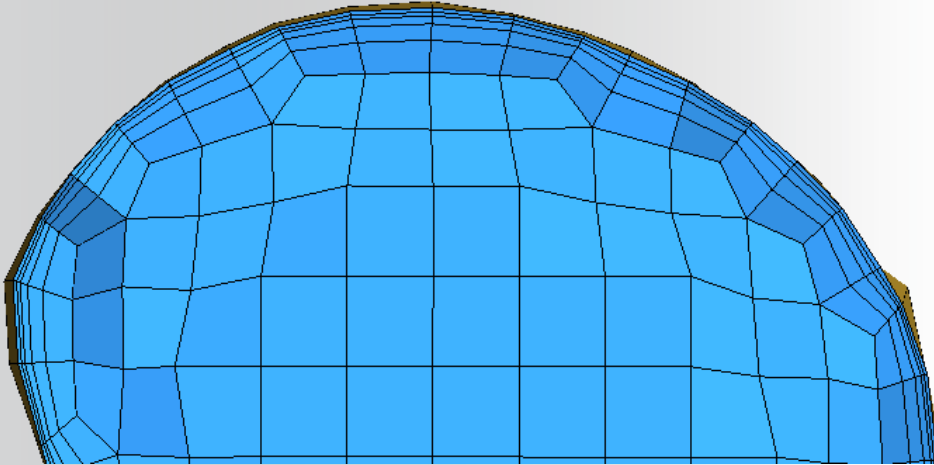
- In the model tree, **Right Click** on **Mesh**
- Select **Cut Plane -> Manage Cut Plane**

- Set the **Method** to “**Middle Y Plane**”
- In the Model Tree, turn on **Mesh -> Volume**
- Adjust the **Fraction Value** slider to cut thru different parallel slices.
 - Try out some other Methods
- When you find a cross section you want to “keep”
 - Click **Create mesh subset** at bottom
 - You can view these later thru the **Mesh** subsets in the model tree



Further Investigation

- For those that have the time, the following pages introduce additional concepts



Split Inflation Layer

- Prism should be selected for any surface parts that are curvy and those which are flat, but not aligned with the XYZ coordinate system
 - Without the inflation layer, elements at the surface will be low quality or inverted, so this is necessary even for a structural analysis.
 - Since only one inflation layer is made, it is useful to split the layer for CFD analysis
 - **Edit Mesh > Split Mesh > Split Edges**

Edit Mesh

- Click ON **Propagate**



- You should be in cutplane mode or viewing a subset of internal mesh
- Keep the default **Split ratio** of 0.5, which splits in the middle
- Press the **Select edge(s)** button  and select a normal edge, then middle click
- Continue splitting until you achieve the first layer height you want

