⌨ Shortcut Keys

Mesh selection shortcuts

- While using the paintbrush tool, hold ALT and left click & drag to adjust paint brush size
- Hold CTRL + Click & drag over areas to DESELECT selected areas
- Hold SHIFT + Click & drag over areas to SELECT additional areas

Mesh rotation shortcuts

- Hold CTRL + Right Click and drag to PAN
- Right click and drag to rotate mesh
  - Closer to mesh= 3 dimensional rotation
  - Further from mesh= rotation on a plane

Mesh viewing shortcuts

- Select the open eye next to any object on the left hand side of the screen to hide. See next for a faster way to hide meshes
- If you have multiple meshes open and want to view one or more meshes ONLY:
  - For one mesh: Right click the mesh on the left hand side of the screen, choose "Show Selected Only"
  - For multiple meshes: Holding SHIFT, select the meshes you want to view, then right click a selected mesh and choose "Show Selected Only"
- To change the color of your mesh: On the right side of the screen under "Properties" Select the colored box next to "Material", select the box with "..." and choose a new color for your mesh

General shortcuts

- CTRL + Z to undo action

🫁 Geomagic Anatomy

Toolbar

Tree

- left side of screen, shows loaded meshes, planes, vectors etc.

- If missing, right click on the toolbar, click on tree to turn on visibility

Properties

✂️ CUTTING OUT MOLDS

Step 1: Getting Started

1a: Getting Started to Work Remotely <u>with Remote Desktop</u>

1. Schedule time in AISOS
2. Plug in your external HD (to AISOS computer)
3. On your personal computer, Connect to the university VPN with Cicso using your X500
4. On your personal computer, open the "Remote Desktop Connection" app,

    Remote Desktop Connection App

   a. Follow the instructions on this page to connect to an AISOS computer
      - Here is the document with the computer IDs and login information
5. Once the AISOS computer loads on your screen, Click geomagic Design X to open

   a. Close the pop-up window asking you to update the software
6. Open your folder where mesh files are stored
7. Open the folder with mold/bone photos for reference

   a. You may also want the molds in hand to inspect (if your project has molds)

    Design X 2024.1
8. Open the google sheet to record the date you have cut out the mesh
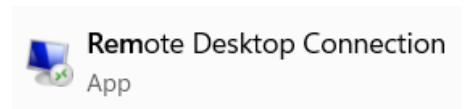
Step 1b: Getting Started to Work Remotely <u>on your computer</u> (REQUIRES SOFTWARE DOWNLOADS TO YOUR COMPUTER)

1. Download the geomagic software
2. On your personal computer, Connect to the university VPN with Cicso using your X500
3. Click geomagic Design X to open

   a. Plug in the information for the floating license
   b. Close the pop-up window asking you to update the software
4. Open your folder where mesh files are stored

5. Open the folder with mold/bone photos for reference
    a. You may also want the molds in hand to inspect (if your project has molds)
6. Open the [google sheet to record the date](#) to record the date you have cut out the mesh

Step 1c: Getting Started to Work in Heller 431

1. Schedule time on the AISOS calendar for the computer you will be using
2. Plug in your external HD
3. Click geomagic Design X to open
    a. Close the pop-up window asking you to update the software
4. Open your folder where mesh files are stored
5. Open the folder with mold/bone photos for reference
    a. You may also want the molds in hand to inspect (if your project has molds)
6. Open the [google sheet to record the date](#) you have cut out the mesh
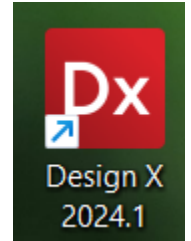
☐ Step 2: Importing mesh into geomagic

1. Either click the import button (yellow folder with red arrow pointing down on the top left of the geomagic software), OR click and drag the file into the geomagic workspace
    a. Import the FUSED version of the mesh. (Import Only)
    b. **Change the measurement from M to MM.**
2. Make sure you copy the original mesh file name from the folder of origination
    a. Slowly double click on the name of the new mesh (should be "Default") in the tree, paste in the name of the original file
        ■   **The mesh will save the original file name if it is not an obj, skip this step if it's a .ply!**
3. Click the MENU tab, File, Save As, to **save your geomagic project** to access later if you need to stop before identifying all modifications on the bone.
    a. Create a folder in the Harddrive if there is not already one titled **Geomagic Projects** then save your project there. You can have multiple meshes in one geomagic project just make sure you save as you go
        ■   If single mesh, title the project the specimen ID and mold ID
        ■   If multiple meshes, id call it your name, mesh cleanup or something appropriate
        ■   **GEMOMAGIC DOES GET GRUMPY.** I wouldn't do more than 10-30 meshes in one file, 30+ may will run slow and may crash
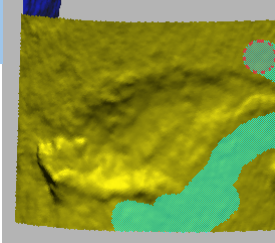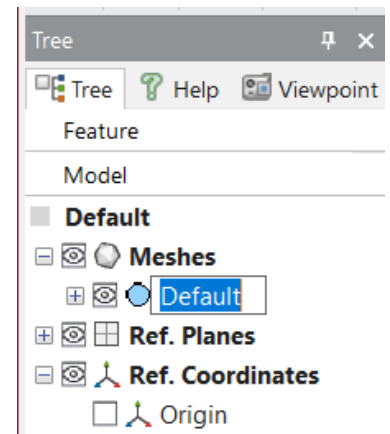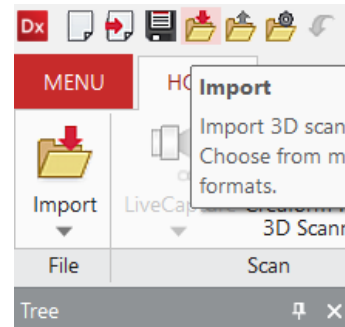
- **SAVE OFTEN**

## 🏞️ Step 3: Identifying areas of interest

1. Change the view from ___ to [   ]. This will show you the changes in topography as colors. Extremes are in deep blue and bright red.
2. Click on the range of colors to allow for editing, select the lower arrow and drag up. This will decrease the threshold for the lower values and give you a clearer view of where the tooth mark meets the bone surface.
3. Identify the deepest blue parts where the topography flattens out, this is your tooth mark outline. You should be able to follow that blue line all the way around with your eyes before making any edits
   a. **WHEN IN DOUBT, CONSULT THE PROJECT OVERLORD**

✂️ Step 4: Cutting out the modification

1. If you have multiple meshes in your geomagic project, right click on the mesh you're working on in the Tree and click **Show Selected Only**

   a. **The paintbrush will select ANY mesh on the plane you are selecting so do not have any meshes visible except the one you are working on!**

2. Choose the paintbrush tool from the toolbar. Left click and hold + drag to highlight parts of the mesh you wish to remove.

   a. **TO ADD MORE SELECTED DATA without erasing your previous selection:** hold down the shift button to select additional parts of the mesh

   a. **TO UNDO PART OF YOUR SELECTION:** If you accidentally selected an important part of the mesh, hold CTRL and click the parts you don't want highlighted to remove it.

   b. **TO CHANGE BRUSH SIZE:** Hold ALT + click & drag to change paint brush size.

   c. Shortcut Keys

2. Click the delete button on the keyboard when you are ready to remove parts of the mesh

   a. CTRL + Z to undo

   b. If the software asks you if you are sure you want to also delete your entire mesh, CLICK NO, This will delete your whole mesh!

🆘 Step 3 & 4: HELP

1. HELP! Geomagic is trying to delete my entire mesh
   a. Holding CTRL, click the highlighted mesh on the left hand side to deselect
2. CTRL + Z to undo!
3. I LOST THE TREE!
   a. Don't fret, right click on the toolbar at the top where you can find the square selection tool and the paintbrush, and click "Tree"
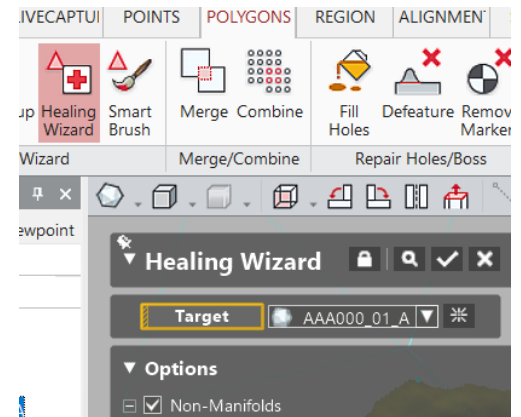
🧽 Step 5: Cleaning up the mesh

1. In the polygons tab, select "Healing Wizard", the target should be the mesh you are working on. It might not be if you have more than one mesh selected or visible.
   a. If the wrong mesh is selected, select the star to remove the mesh from the target and select the mesh you want from the tree
   b. If there is nothing to clean, the check mark will be unclickable
2. Click the check mark to run the healing wizard.
   a. This will remove any floaters
   b. **If the check mark cannot be clicked it is because your mesh is clean, congrats!**
3. If your mesh has some small holes, navigate to the polygons tab and select "Fill Holes"
   a. The target should be the mesh you wish to have holes filled
   b. Click and drag on the screen so that you are selecting the entire mesh. This will highlight any holes that needs filling
   c. Your mesh outline will be highlighted in blue YOU DO NOT WANT THIS. Click on the mesh outline to deselect that from the hole filling process
      i. If you fill holes with the outline selected it will fill in the entire mesh which is not what we want. If you do this just tap ctrl + Z and start over
4. Repeat steps 1-2 (apply healing wizard) after filling any holes before moving on to step 5
5. In the polygons tab click the "Global Remesh" button to properly redistribute the triangles across the mesh
   a. Edge length multiplier should be 1, this automatically calculates the Target Average Edge Length for you
   b. Make Clean And Manifold Solid Mesh should be **UNCHECKED**
6. In the polygons tab, select "Edit boundaries", click on the edge of your mesh (it should light up red all the way around), click the check mark to complete the action
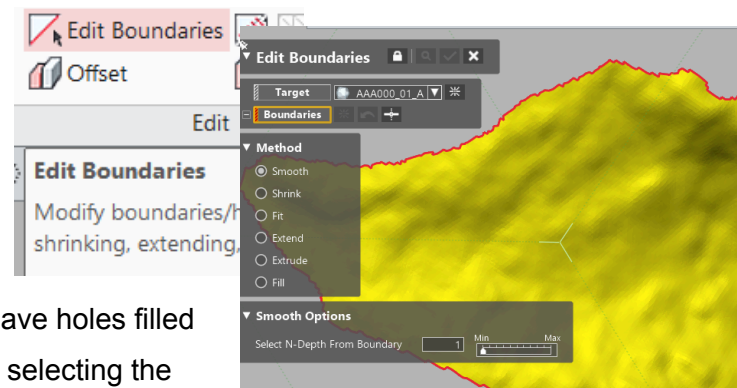
a. Method should be set to smooth

b. Smooth options should be 1

7. **REPEAT STEPS 3-5 UNTIL ALL MODIFICATIONS ON THE BONE HAVE BEEN IDENTIFIED**

☐ Step 6: exporting your cut-out mesh

- Yay you did it!

1. Select your cut out modifications from the tree, Click the export button (yellow folder with gray arrow pointing up)

    a. First mesh, hold shift, click last mesh to select all

2. The "Entities" should be the meshes you wish to export, select the arrow to proceed with exporting

3. Navigate to your folder and save this new mesh with the new name you have given it.

4. Change the "Save as type:" to "CyberWare ASCII File (*.ply)

5. Click the box "Export Each Scan Data As Separate File"

6. Click the box "Use Scan Data Name"

    a. This will maintain your naming scheme relieving you of having to rename everything, yay!
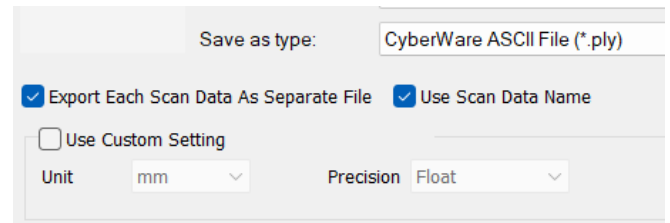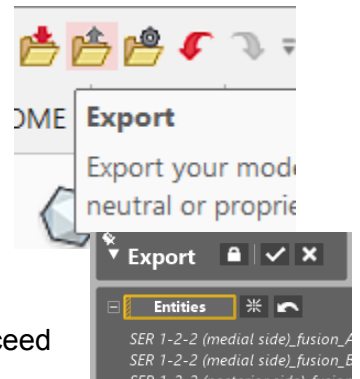
🚀 Step 7: REPEAT!

1. If you are wanting to clean multiple meshes before aligning, return to Step 2

2. If you wish to complete all steps mesh by mesh, continue to next section

---

ALIGNING MESHES

Step 1: Import all the cutout modifications

- Once you have cut out EVERY modification from ALL scans you are ready to get your data ready for processing!

1. Click the button to import meshes, navigate to your folder with all the cut out meshes and import all of them

    a. ctrl + A, or

b. click the first in the list, hold select and scroll to the bottom of the list and select the last mesh in the list

2. Save this as a new workspace titled to match the folder name relating to your project
    a. FOLDER: For <u>non-landmarking</u> projects that are working with **mold scans** should be titled "cut out modifications_aligned_undecimated"
    b. FOLDER: For <u>non-landmarking</u> projects that are working with **bone scans** should be titled "cut out modifications_aligned_undecimated_unfixed normals"
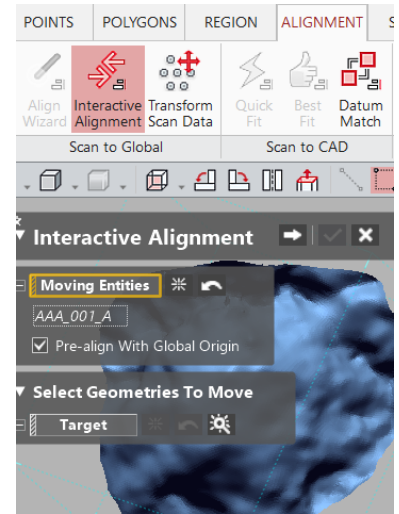
Step 2a: Aligning your meshes to origin

1. Select **all meshes** in the meshes you just imported, navigate to the alignment tab and select interactive alignment
    a. All the selected meshes should be listed under "Moving Entities"
    b. The "Pre-align With Global Origin" box should be checked
2. Click the Right pointing arrow within the "Interactive Alignment" window to progress
3. The very bottom box labelled Target should be set to "Coordinate" and the dropdown should be set to "Origin"
   a. Ignore the middle box labeled "Moving" for now
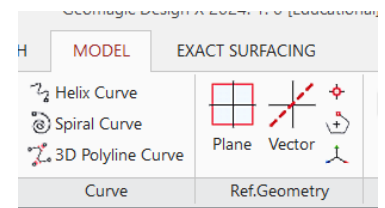
Step 2b: Getting your meshes in the same orientation

1. Make sure you are on the box selection tool and not the paint selection tool
2. **Add a plane**
    a. In the Model Tab, under the "Ref. Geometry" section, select "Plane"
    b. If there is something listed under entries, clear it by clicking the star
    c. Select the outline of your mesh to extract a plane
        i. Do not remove outliers
        ii. Sampling ratio 100%
        iii. YES axis constraint
            1. Use Specified Direction As Initial Guess
    d. Click the check mark to apply the new plane
3. **Add a vector**
    a. In the Model Tab, under the "Ref. Geometry" section, select "Vector"
    b. Clear the entities as it will likely be the new plane.
    c. Either Select your mesh from the tree or, select the paint tool (flood selection mode) from the toolbar and select your mesh
        i. Method:Extract
        ii. Remove Outlier: NO
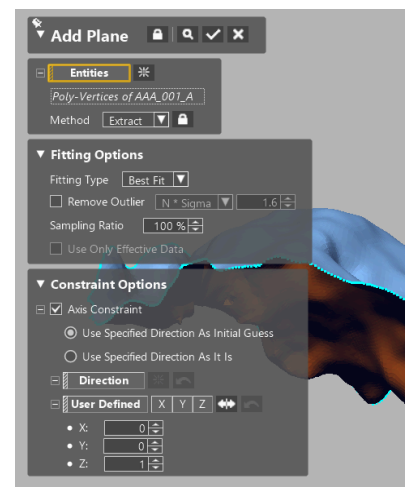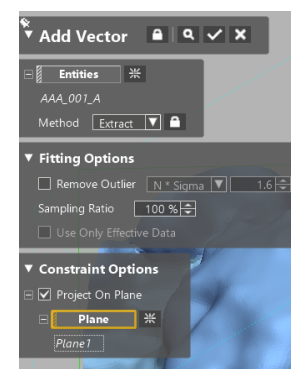
   iii. Sampling Ratio: 100%

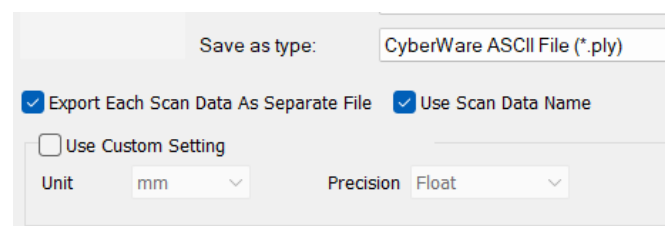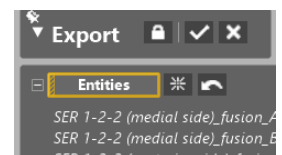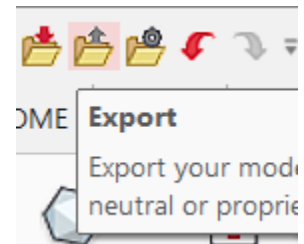   **iv. Project on Plane: YES**

    1. Plane from #2 above!

4. **Interactive alignment**
   a. Navigate to the alignment tab and select interactive alignment
      i. The mesh with the new plane and vector should be the only entity
         1. Pre-align with global origin: YES
   b. Click the Right pointing arrow within the "Interactive Alignment" window to progress
   c. In the "Moving" window, Select "X-Y-Z"
      i. Axis Y: Select NEW PLANE
      ii. Axis X: Select NEW VECTOR
   d. In the "Target" window. Select "Coordinate"
      i. To: "Origin"
   e. Click the check mark to align your modification based on its bottom z value and the longest axis
   f. <mark>Once the mesh has been fully aligned, double click on its name in the tree and add "_a"</mark>
   g. **Delete the new plane and vector so you don't confuse yourself**
   h. **SAVE GEOMAGIC WORKSPACE OFTEN**
   i. Repeat steps in <u>8b</u> for ALL cutout meshes
      i. You can oscillate between steps 8b and 9 as you see fit
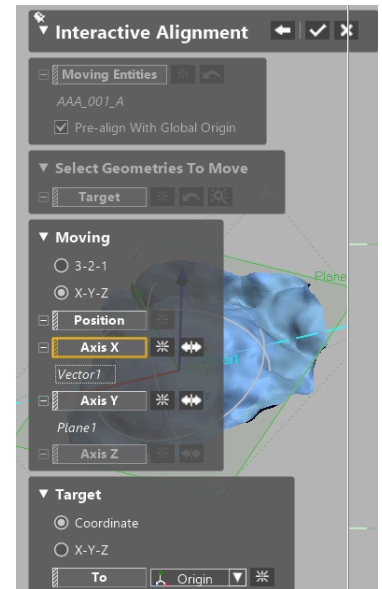
Step 3: Export your aligned, undecimated scans

1. Select your cut out modifications from the tree, Click the export button (yellow folder with gray arrow pointing up)
   a. First mesh, hold shift, click last mesh to select all
2. The "Entities" should be the meshes you wish to export, select the arrow to proceed with exporting
3. Navigate to your folder and save this new mesh with the new name you have given it.
   a. FOLDER: For <u>non-landmarking</u> projects that are working with **mold scans** should be titled "cut out modifications_aligned_undecimated"
   b. FOLDER: For <u>non-landmarking</u> projects that are working with **bone scans** should be titled "cut out modifications_aligned_undecimated_unfixed normals"
4. Change the "Save as type:" to "CyberWare ASCII File (*.ply)"
5. Click the box "Export Each Scan Data As Separate File"
6. Click the box "Use Scan Data Name"

a. This will maintain your naming scheme relieving you of having to rename everything, yay!

Step 4: Recording additional data

1. Right click the first mesh in your list of meshes in the tree, select "Properties"
   a. On the right hand side of the screen, you will see a properties from which you will collect several variables on each mesh
2. Navigate to google drive, open the **google sheet for your dataset**. Locate the modification you currently have highlighted
3. In the properties window, from the **Nominal geometry** section you will <u>record</u>:
   a. **Bounding Box** Width, Height, and Depth
      i. Click the dropdown arrow to see which value corresponds to which variable
   b. **Poly-Faces Count**
   c. Calculate and record the **volume**
   d. Calculate and record the **area**
   e. <mark>To record each one easily, click on the individual box with the value of interest, hold <u>ctrl</u> and press c on your keyboard, click on the cell in your spreadsheet where you want the number to go, hold <u>ctrl + shift press v</u></mark>
      i. <mark>**OR**</mark> <u>OR</u> press windows key + v to open copy clipboard while in software that is not geomagic to ensure the clipboard is ON, go back to geomagic and copy every value in order. Return to your data sheet, press windows key + v, scroll to the value matching the correct variable, click on the three dots and choose the option to paste as text

Step 5: Decimate Triangles (DTA Projects ONLY)

1. For DTA to work properly, the meshes cannot be more than 10,000 triangles (poly-faces count) and ideally every mesh should be made up of the same number of triangles.
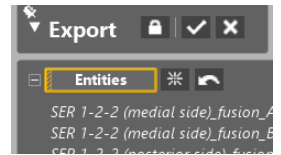2. Change your status to STUCK and follow up with the project overlord to decide what mesh size is appropriate
3. <mark>**ONCE YOU DECIMATE, CHANGE THE NAME BY DOUBLE CLICKING ON IT IN THE TREE TO ADD "_d"**</mark>

Nominal Geometry

Bounding Box		3.314, 4.9685, 4.5342
	Width	3.314 mm
	Height	4.9685 mm
	Depth	4.5342 mm
	Center Position	33.816, 12.2853, 26.5252
	Poly-Vertex Precision	Float
	Poly-Faces Count	13,801
	Poly-Vertices Count	7,187
	Boundaries Count	1
	Approx. Volume (49%...	0.347 mm³
	Area	8.0871 mm²
Mass Center		33.6547, 12.2946, 26.48...
	x	33.6547 mm
	y	12.2946 mm
	z	26.4818 mm
	Scanner Type	Undefined

Step 6: Save your aligned, decimated scans

1. Select your cut out modifications from the tree, Click the export button (yellow folder with gray arrow pointing up)
   a. First mesh, hold shift, click last mesh to select all
2. The "Entities" should be the meshes you wish to export, select the arrow to proceed with exporting
3. Navigate to your folder and save this new mesh with the new name you have given it.
   a. FOLDER: For <u>non-landmarking</u> projects that are working with **mold scans** should be titled "cut out modifications_aligned_decimated"
   b. FOLDER: For <u>non-landmarking</u> projects that are working with **bone scans** should be titled "cut out modifications_aligned_decimated_unfixed normals"
4. Change the "Save as type:" to "CyberWare ASCII File (*.ply)
5. Click the box "Export Each Scan Data As Separate File"
6. Click the box "Use Scan Data Name"
   b. This will maintain your naming scheme relieving you of having to rename everything, yay!