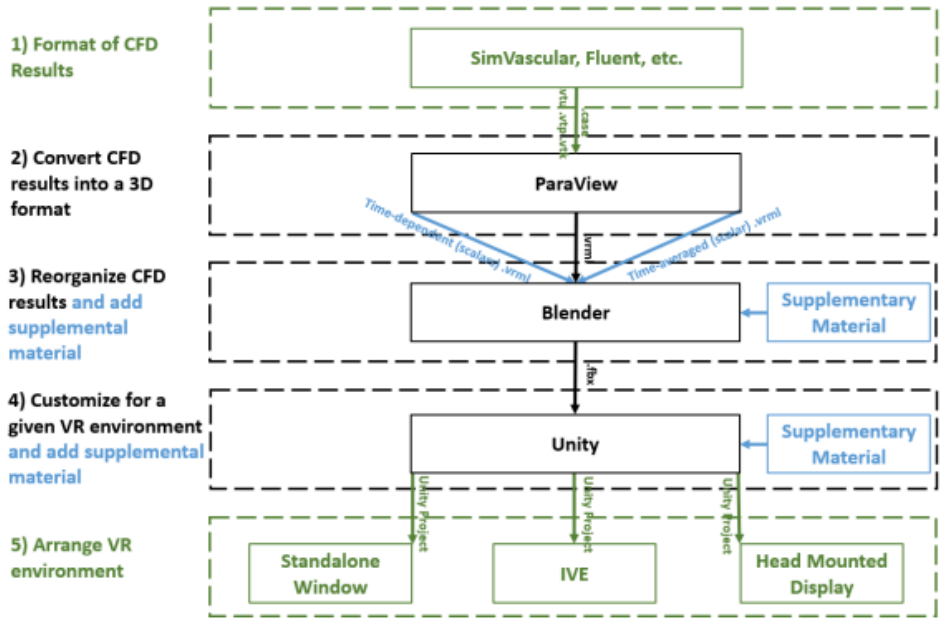
Yellow denotes the user needs to do this step

Green denotes that the user needs to pick one of the options

Blue denotes that this step is optional

No highlight means that it is just the title of a section (there are no specific steps in the manuscript)

Contents



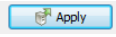
1 Download necessary programs

1) Download ParaView
    a. https://www.paraview.org/download/
2) Download Blender
    a. https://www.blender.org/download/
3) Download Unity (Version 2017.#(Doesn't matter))
    a. https://store.unity.com/download?ref=personal
4) IF USER WOULD LIKE TO VIEW VOLUMETRIC IMAGING DATA
    a. Download Image J
        i. https://imagej.nih.gov/ij/download.html
    b. Download VolView
        i. https://www.kitware.com/volview/

2 Convert CFD results into file formats that store 3D geometry

2.1a Convert CFD results into Streamlines using ParaView

Below, are the specific commands to create velocity streamlines. Icons are identified with **bolded** font, keyboard commands are underlined., and keyboard entries are *italicized step number* (i.e. *(1)*).

1. Load CFD Project
    a. Open ParaView
    b. If your CFD simulation has multiple cardiac cycles create a new folder that only includes the last cardiac cycle
        i. For example: If you have 3500 Time Steps going over 7 cardiac cycles (500 time steps per cycle) with the number of time steps between restarts as 20 you would select all of the files that are 3020 and greater into a separate folder
    c. Load the CFD results created in step 1.b **(.vtu, or .vtp)** or a **.case** files into ParaView by selecting **File → Open**.
        i. A new window should pop up, select the CFD simulation and Press **OK**
    d. In the properties window press **Apply** *(1)* 

        i. The model should now appear in the Layout Window
    e. Select the Coloring as whatever texture gradient the wall mesh should be (Example: Pressure, wss, or Solid Color) (Figure *1* a)
        i. Select **Choose preset icon** () A new window should pop up, in Preset select jet and press **Apply** then press **Close** (Figure *1* b).

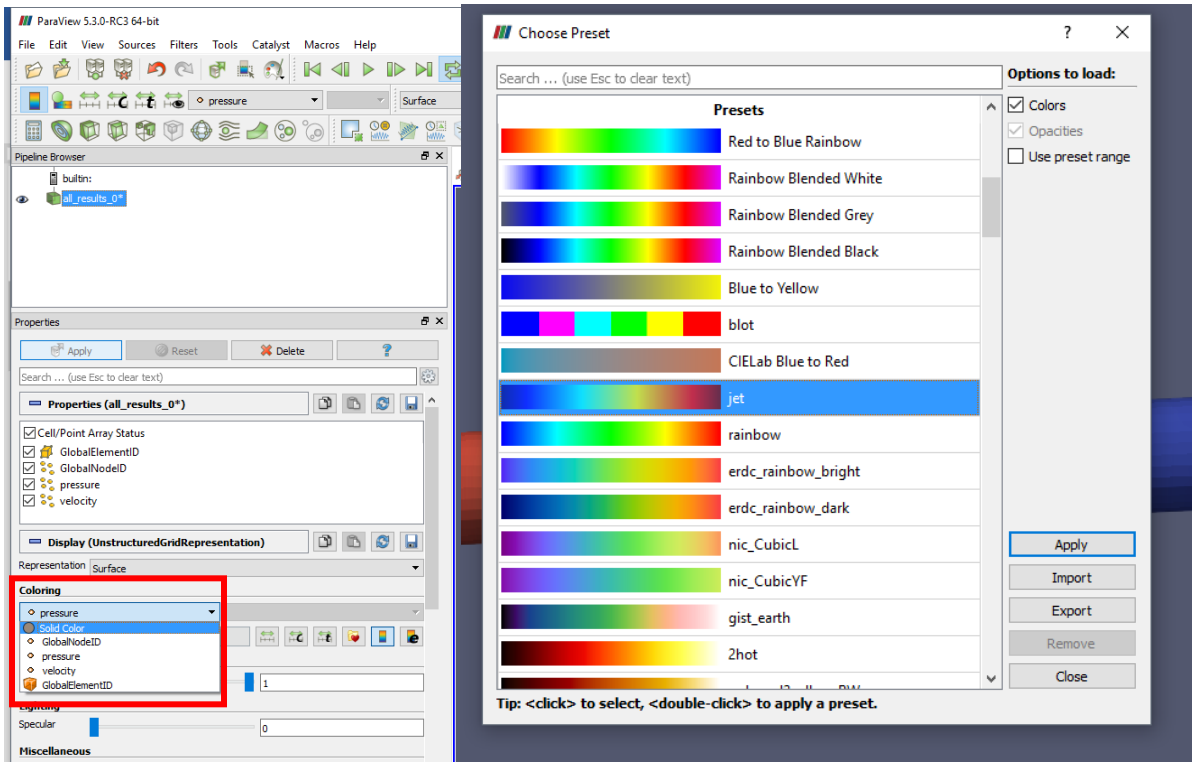


Figure 1: a) select the wall value you would like to show for each time step. b) Choose Present jet scale that should be used for all of your scaled

ii. If you are going to select **pressure** for each time step the scale needs to be set/adjusted

1. Select the **Rescale to data range over all timesteps** () in the Color Map Editor (Figure *2* a)
    a. A new Window should pop up where then the user should then select **Rescale and disable automatic rescaling** (Figure *2* b) or in an older version of ParaView click **Yes**

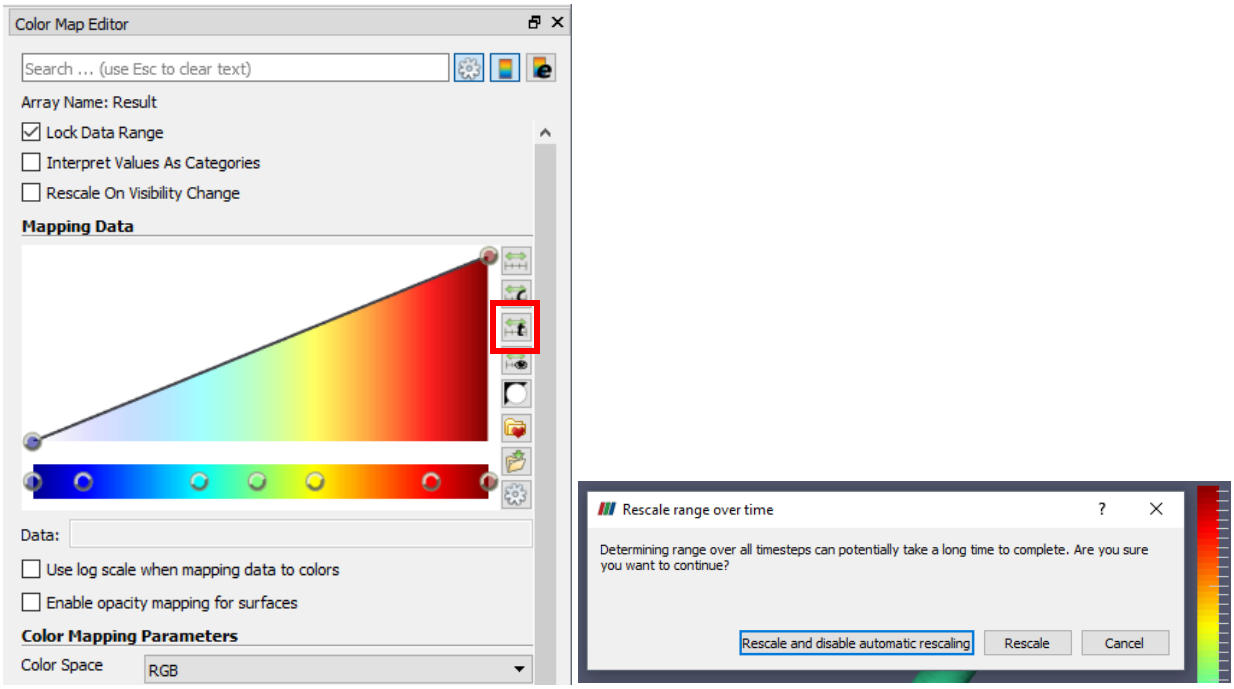


Figure 2: a) Color Map Editor b) setting range of scale

2. Create Coordinate Vector Velocity- Not always necessary
    a. If Velocity is not given but, x_velocity, y_velocity, and z_velocity where (*Figure 3* a), scalar velocities must be converted to a coordinate vector velocity.
    b. Click the **calculator icon** () (*Figure 3* b); type in iHat*velocity_X+jHat*velocity_Y+kHat*velocity_Z.
        i. Note, if user's scaler velocity information was x_Velocity, y_Velocity, z_Velocity the user would type in: iHat*x_Velocity+jHat*y_Velocity+kHat*z_Velocity.
    c. In the **Result Array Name,** type in Velocity.2 and press **Apply** *(2)*.
        i. An accurate representation of coordinate vector velocity is now depicted in the CFD model.
    d. If Velocity is already provided, the previous step should be skipped.

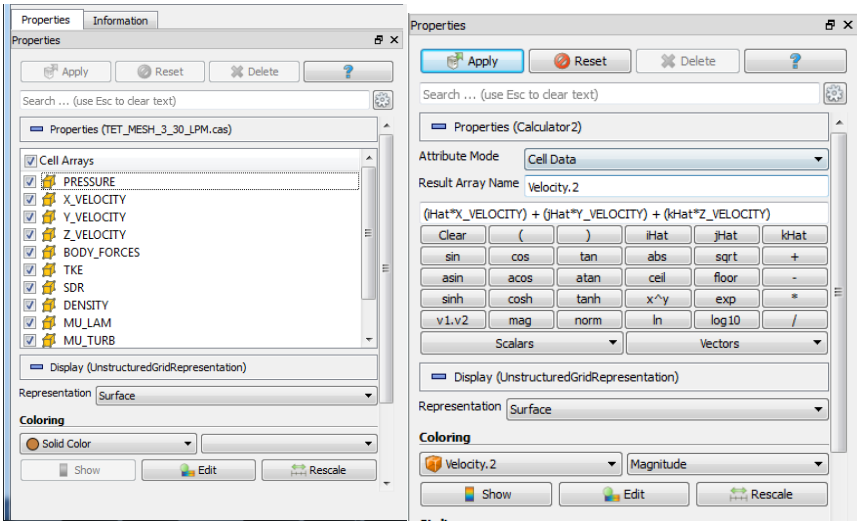


Figure 3**:** a) Example (without velocity) of CFD results after loading the CFD file into. b) Calculator icons interface with the velocity equation and the Results Array Name.

3. Create Streamlines
    a. To create streamlines representing flow and velocity profile, click on the **streamline tracer icon** ( ) and in the streamline tracker interface (*Figure 4* a).
        i. Select **Vectors** as Velocity and **Integration Direction** as FORWARD. Select **Seed Type** as Point Source. Select the desired **Number of Points** (streamlines).
            1. In *Figure 4* a, the streamlines are selected at 100, which is a low number of streamlines. Before pressing apply, there is a white wireframe tube with an asterisk is in the middle. Move this asterisk into the center part of the models inflow (*Figure 4* b), and set the radius of the wireframe circle (in **Sphere Parameters – Radius**) to roughly the radius of the inlet geometry then press **Apply** *(3)*.
        ii. Select Solid Color as **Coloring** (Figure *4* a)

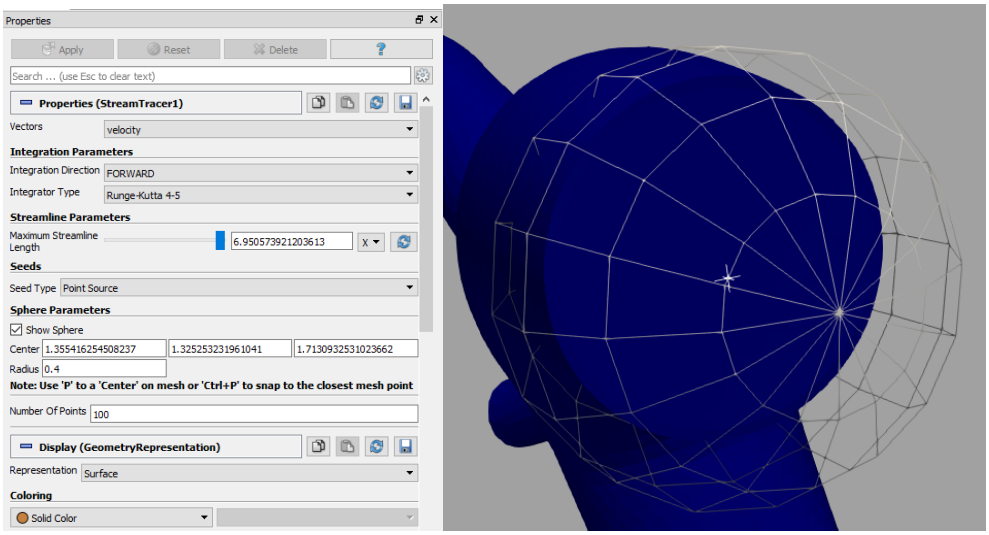


Figure 4**:** a) Streamline tracer icons interface with the Vector, Integration Direction, Radius and Number of Points selected as Velocity, FORWARD, 0.4, and 100. b) Wireframe circle roughly matches the radius of the inlet and the asterisk in the center part of the CFD models inflow.

4. Add Volume to Streamline
    a. To add volume to the streamlines, select **Filters → Alphabetical → Tube** on the top toolbar.
    b. Adjust the **Radius** of the desired tubed-streamlines, for the **coloring**, select Velocity (see *Figure 5*). Then press **Apply** *(4)*.
        i. This step creates the 3D geometry needed for VR and creates the vertex color information indicative of velocity magnitude.

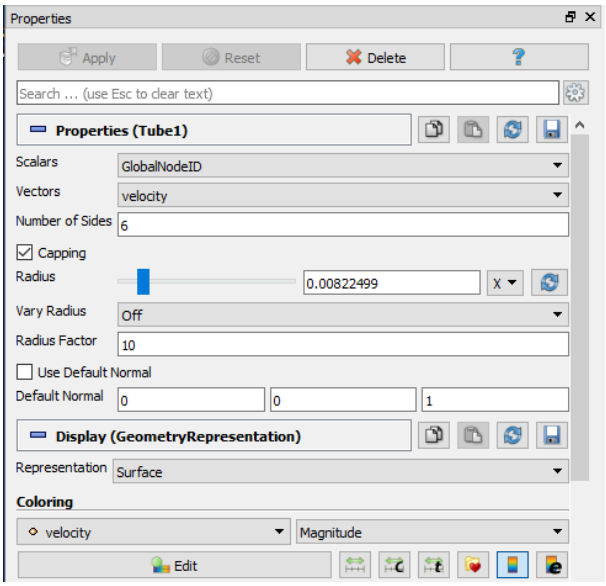


Figure 5**:** Tube interface with Radius set as 8.6297e-05 and the Coloring selected as Velocity.

c. Select **Choose preset icon** (). A new window should pop up, in Preset select jet and press **Apply** then press **Close** (Figure *1* b).
d. Select the **Rescale to data range over all timesteps** () in the Color Map Editor (Figure *2* a)
    i. A new Window should pop up where then the user should then select **Rescale and disable automatic rescaling** (Figure *2* b) or in an older version of ParaView click **Yes**
e. Record the min and max Velocity information for later use in Unity

5. Run Python Script
    a. To run the python script, create a new folder on the desktop and name it "ParaView_vrml".
    b. Locate the ParaView_export.py python script
    c. Open up the ParaView_export.py file
        i. On line 8, change the number to the end time step of the simulation.
        ii. On line 20, change the location of the ParaView_export.py file
            1. For example, the line of code is originally this:
                a. fname = "C:/users/9376vennj/Desktop/ParaView_vrml/foo_res" + format(x) + "_velocity.vrml"
                b. Only change the highlighted part
    d. Save the ParaView_export.py file.
    e. In ParaView, select **Tools → Python Shell** on the top toolbar.
    f. In the Python Shell window, select **Run Script** *(5).*
        i. A new window should pop up, locate the ParaView_export.py file and select run **OK**. (Figure *6*)
        ii. In the main ParaView window, the CFD model should be going through each desired time step.

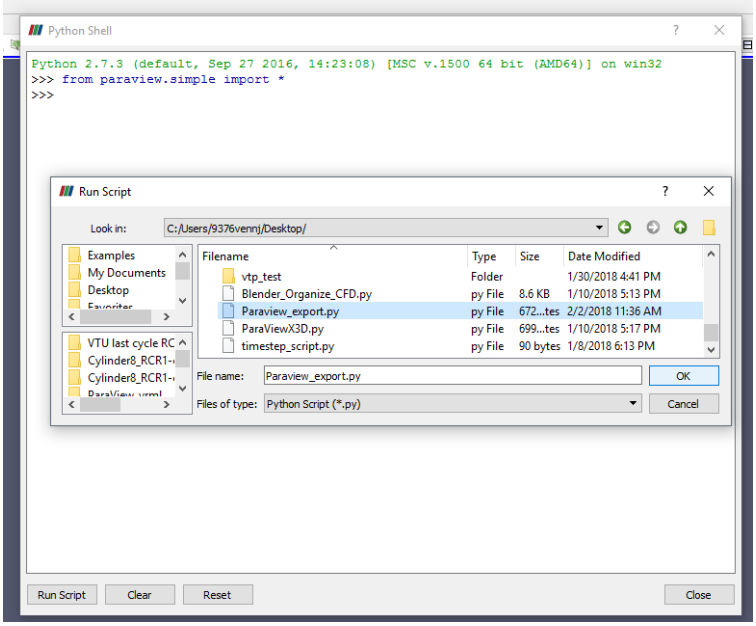


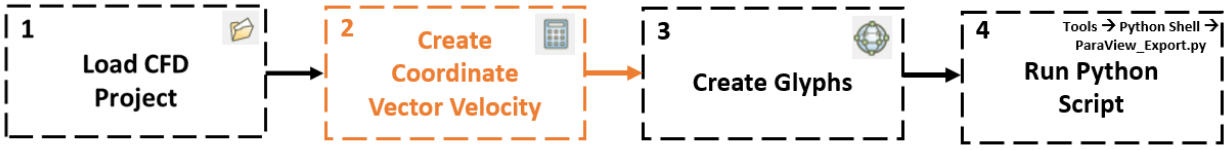
Figure 6: Python Shell window

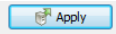
g. After the Python script is complete, go to the ParaView_vrml file, in the file there should be the desired number of time steps in .vrml form.

This completes the steps to convert CFD results into 3D format if the user would like to depict his or her CFD results using streamlines.

2.1b Convert CFD results into Glyphs using ParaView

Below, are the specific commands to create velocity glyphs. Icons are identified with **bolded** font, keyboard commands are <u>underlined</u>., and keyboard entries are *italicized step number* (i.e. *(1)*).



1. Load CFD Project
    a. Open ParaView
    b. If your CFD simulation has multiple cardiac cycles create a new folder that only includes the last cardiac cycle
        i. For example: If you have 3500 Time Steps going over 7 cardiac cycles (500 time steps per cycle) with the number of time steps between restarts as 20 you would select all of the files that were 3020 and greater and put them in a separate folder
    c. Load the CFD results created in step 1.b **(.vtu, or .vtp)** or a **.case** files into ParaView by selecting **File → Open**.
        i. A new window should pop up, select the CFD simulation and Press **OK**
    d. In the properties window press **Apply** *(1)* 
        i. The model should now appear in the Layout Window
    e. Select the Coloring as whatever texture gradient the wall mesh should be (Example: Pressure, wss, or Solid Color) (Figure *7* a)
        i. Select **Choose preset icon** () A new window should pop up, in Preset select <u>jet</u> and press **Apply** then press **Close** (Figure *7* b).

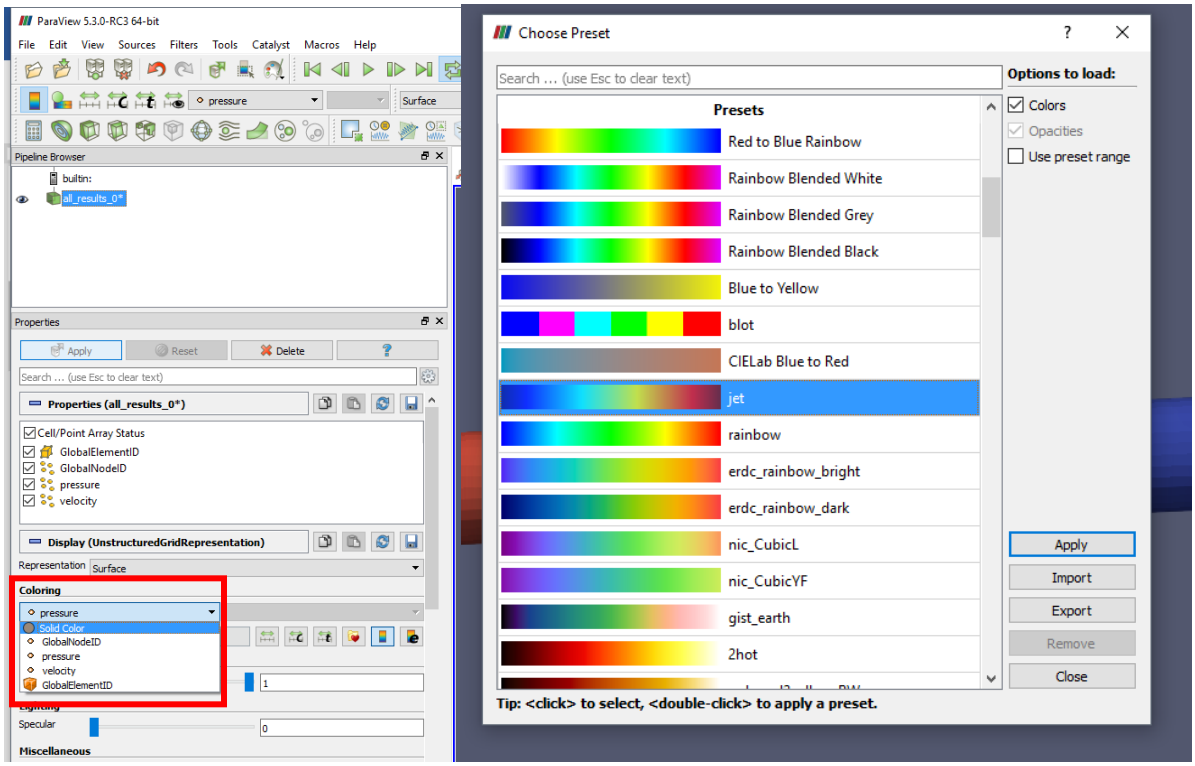


Figure 7 : a) select the wall value you would like to show for each time step. b) Choose Present jet scale that should be used for all of your scaled

ii. If you are going to select **pressure** for each time step the scale needs to be set/adjusted

1. Select the **Rescale to data range over all timesteps** () in the Color Map Editor (Figure *8* a)
    a. A new Window should pop up where then the user should then select **Rescale and disable automatic rescaling** (Figure *8* b) or in an older version of ParaView click **Yes**

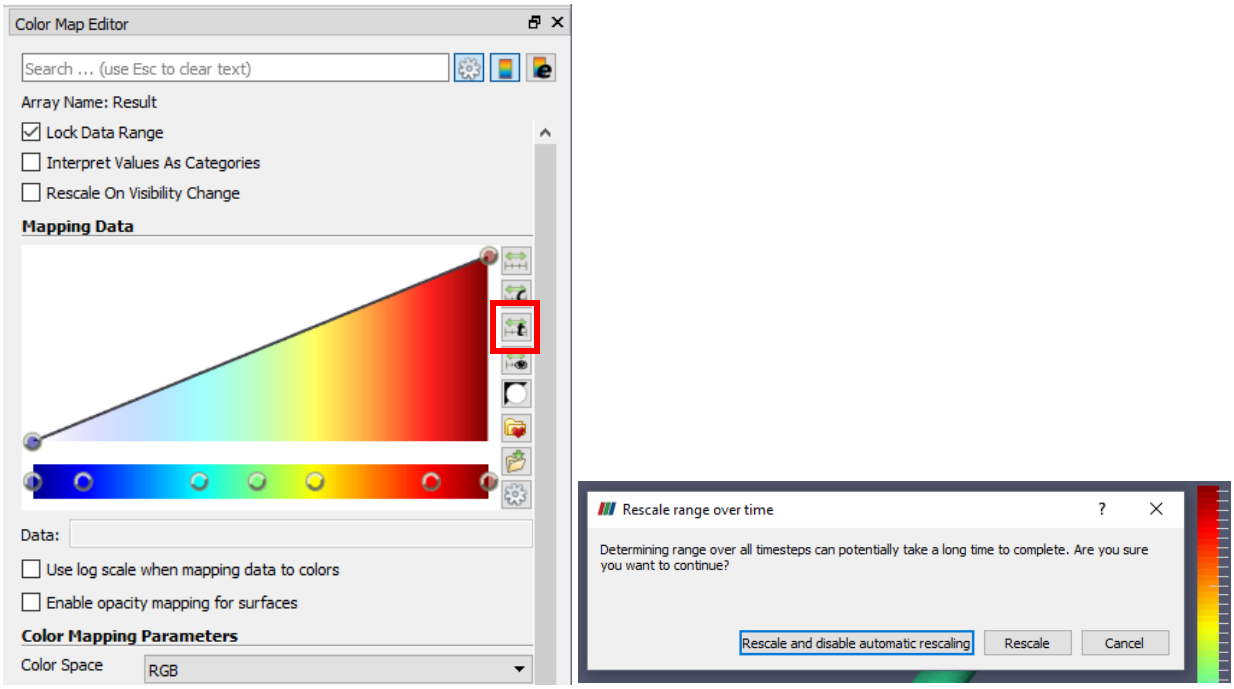


Figure 8: a) Color Map Editor b) setting range of scale

2. Create Coordinate Vector Velocity- Not always necessary
    a. If Velocity is not given but, x_velocity, y_velocity, and z_velocity where (Figure *9* a), scalar velocities must be converted to a coordinate vector velocity.
    b. Click the **calculator icon** () (Figure *9* b); type in iHat*velocity_X+jHat*velocity_Y+kHat*velocity_Z.
        i. Note, if user's scaler velocity information was x_Velocity, y_Velocity, z_Velocity the user would type in: iHat*x_Velocity+jHat*y_Velocity+kHat*z_Velocity.
    c. In the **Result Array Name,** type in Velocity.2 and press **Apply** *(2)*.
        ii. An accurate representation of coordinate vector velocity is now depicted in the CFD model.
    d. If Velocity is already provided, the previous step should be skipped.

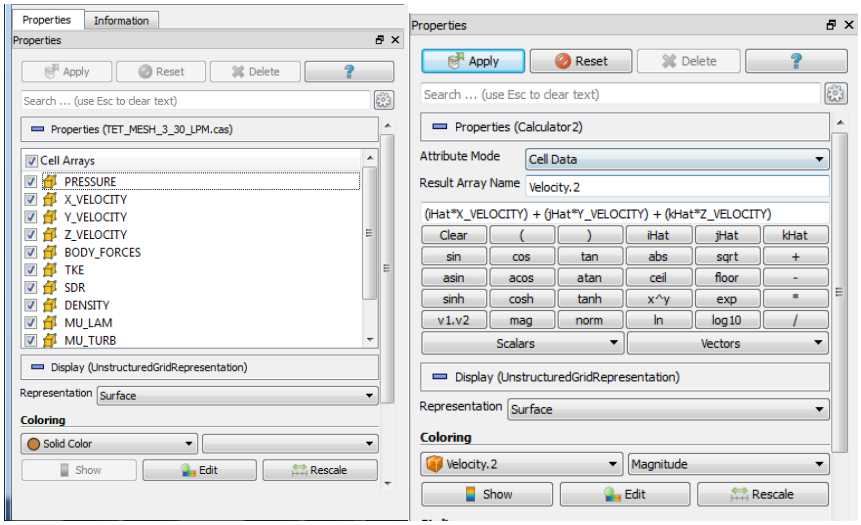


Figure 9**:** a) Example (without velocity) of CFD results after loading the CFD file into. b) Calculator icons interface with the velocity equation and the Results Array Name.

3. Create Glyphs

    a. To create glyphs representing velocity profile, click on the **glyph icon** () and in the glyph interface (*Figure 10* a).
        i. Select **Scalars** as <u>None</u> and **Vectors** as <u>Velocity</u>. Select **Scale Mode** as <u>Vector</u>. Select the desired **Number of Points** (streamlines).
        ii. Select the time step that is peak systole for your simulation (Figure *10 b*)
            1. For example, the simulation used in Figure *10 b* peak systole was at the 9th time step
        iii. In *Figure 10* a, the Scale Factor (size of glyph) and Maximum Number of Sampling points is selected as 0.005 and 5000, this ratio needs to be adjusted depending on the model **Apply** *(3)*.
        iv. NOTE: if the velocity vectors are not in an area you would like (i.e. important model features) – in the Glyph properties masking window adjust the **Speed** until the glyphs are in the desired area
        v. Select **Coloring** as <u>GlyphVector</u>

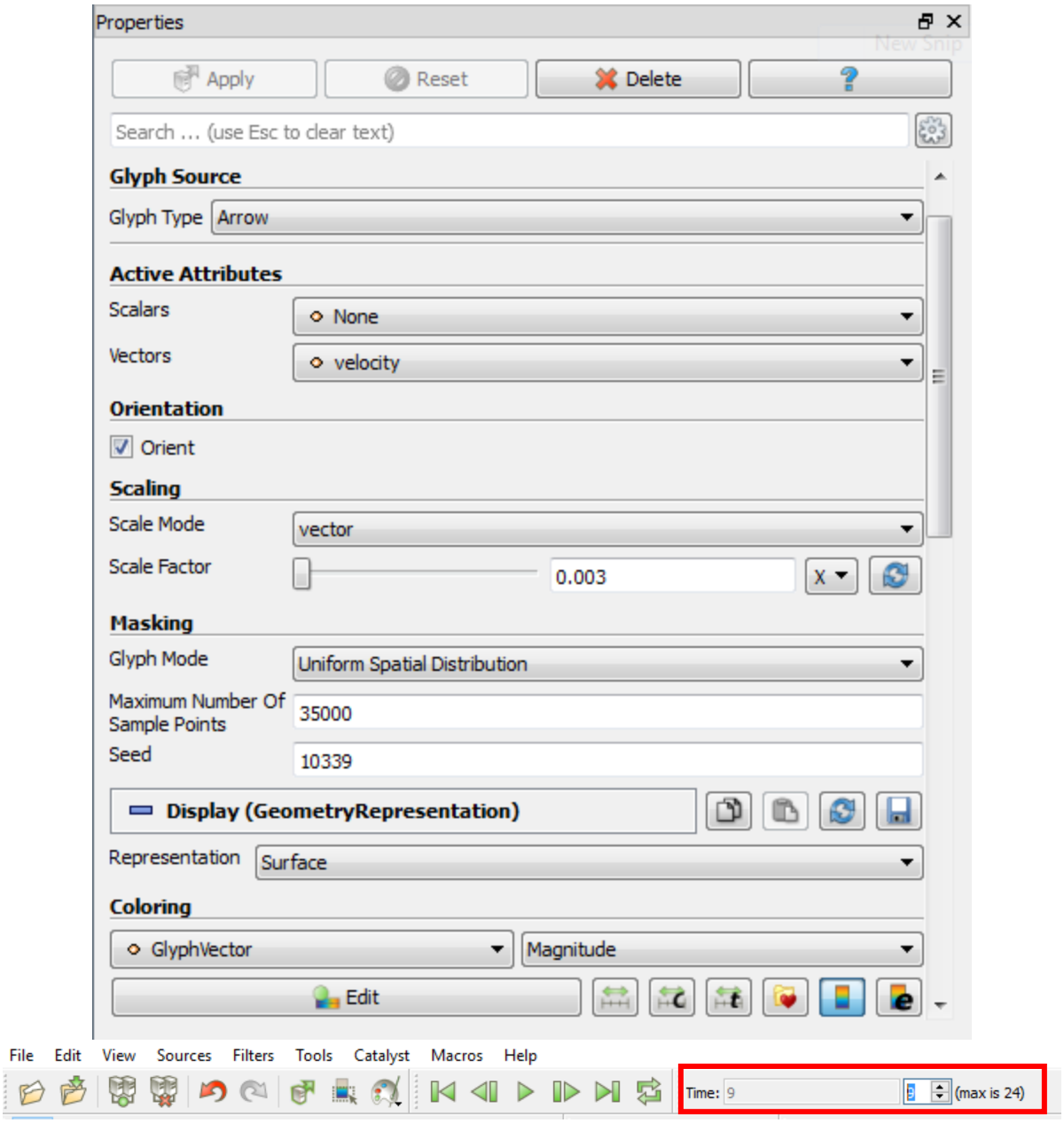


Figure 10: a) Making Glyphs Note that the scalars are set to None, Vectors is set to Velocity, Scale mode is set to Vector, the Scale Factor is very small and Coloring is set to velocity. B) selecting the time step that is peak systole

vi. Select **Choose preset icon** ( ). A new window should pop up, in Preset select jet and press **Apply** then press **Close** (Figure *7* b).

vii. Select the **Rescale to data range over all timesteps** ( ) in the Color Map Editor (Figure *8* a)

viii. A new Window should pop up where then the user should then select **Rescale and disable automatic rescaling** (Figure *8* b) or in an older version of ParaView click **Yes**

ix. Record the min and max Velocity information for later use in Unity

4. Run Python Script
    a. To run the python script, create a new folder on the desktop and name it "ParaView_vrml".
    b. Locate the ParaView_export.py python script
    c. Open up the ParaView_export.py file
        i. On line 8, change the number to the end time step of the simulation.
        ii. On line 20, change the location of the ParaView_export.py file
            1. For example, the line of code is originally this:
                a. fname = "C:/users/9376vennj/Desktop/ParaView_vrml/foo_res" + format(x) + "_velocity.vrml"
                b. Only change the highlighted part
    d. Save the ParaView_export.py file.
    e. In ParaView, select **Tools → Python Shell** on the top toolbar.
    f. In the Python Shell window, select **Run Script** *(5)*.
        i. A new window should pop up, locate the ParaView_export.py file and select run **OK**. (Figure 11)
        ii. In the main ParaView window, the CFD model should be going through each desired time step.

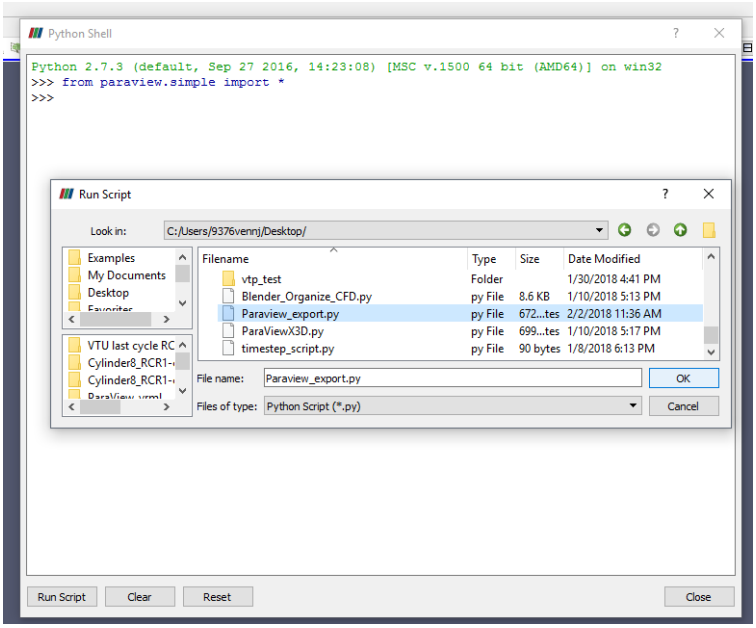
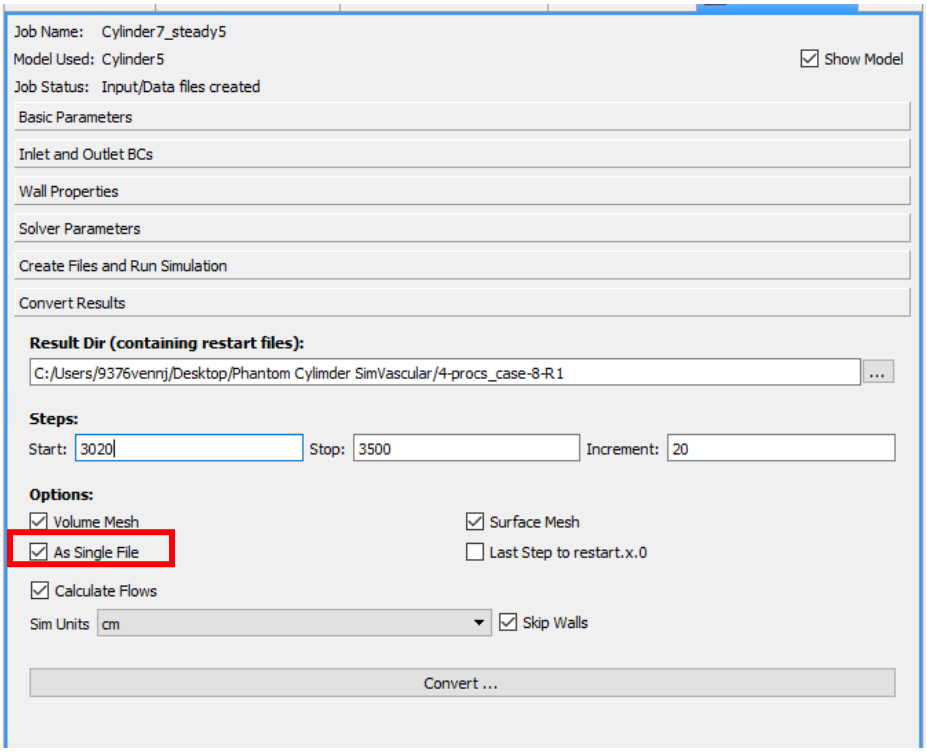


Figure 11: Python Shell window

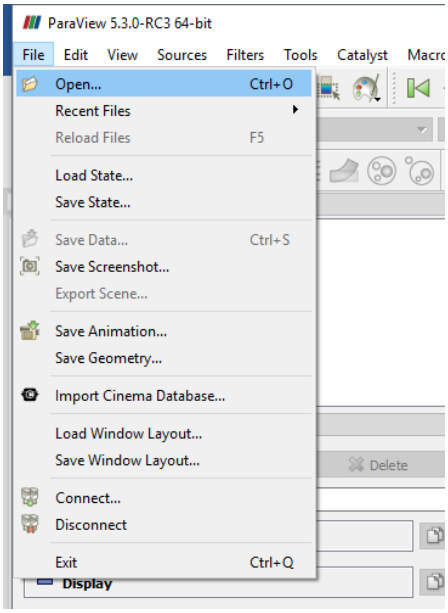
g. After the Python script is complete, go to the ParaView_vrml file, in the file there should be the desired number of time steps in .vrml form.

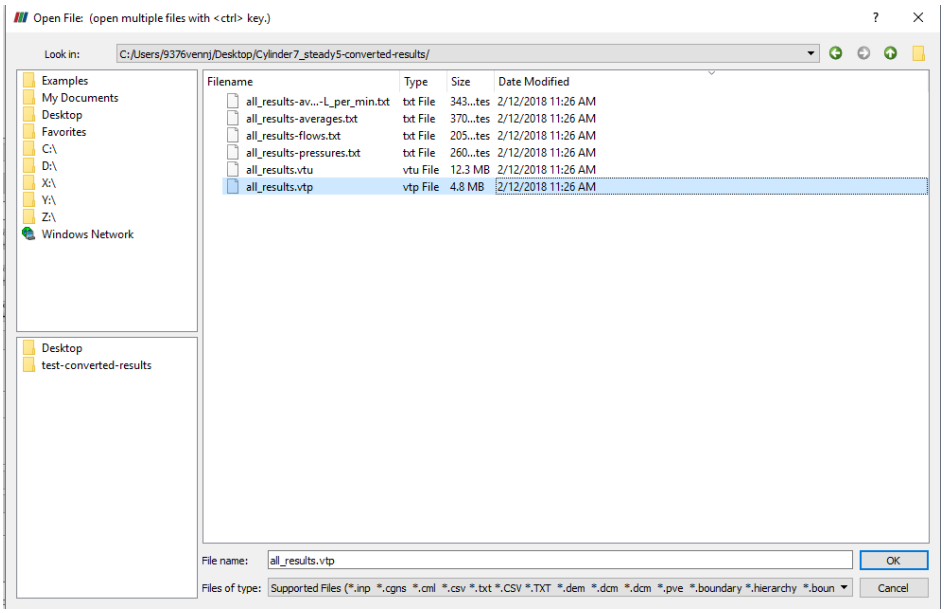
2.2 Convert post-processing results like TAWSS and OSI into a 3D format

1. If using SimVascular when converting your results, select "As Single File" option, select the last cardiac cycle in your simulation and click "Convert…"
    a. For example: If you have 3500 Time Steps going over 7 cardiac cycles with the number of time steps between restarts as 20 you would type in the following

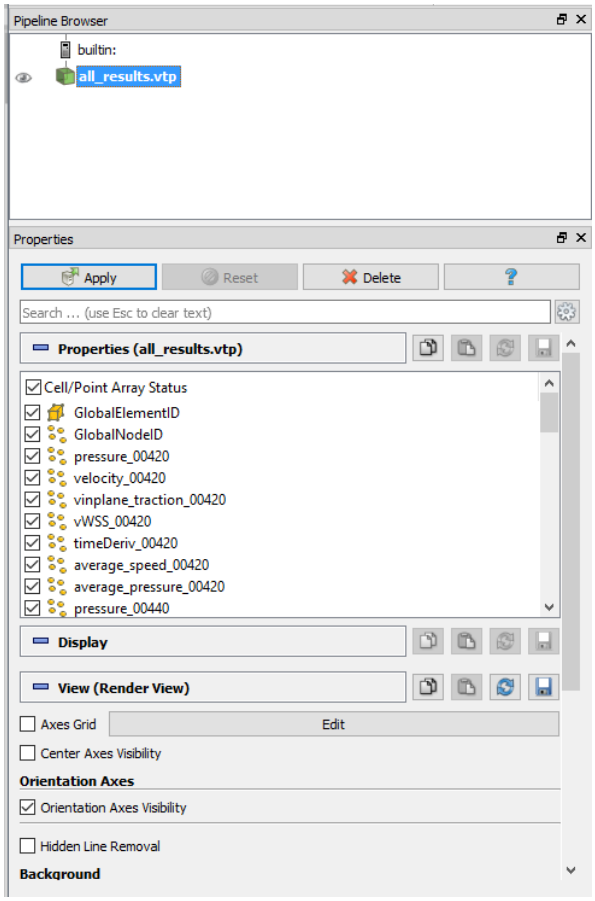


    b. An all_results.vtp and all_results.vtu should be created
        i. .vtp file will be loaded into ParaView
            1. .vtu file is an unstructured mesh file which contains the velocity and pressures at every node in your computational domain. This file will also have WSS computed on the surface
            2. .vtp file s a PolyData file and only contains the solutions on the exterior of your model. It is in this second file that the TAWSS and OSI are computed and stored
2. Open ParaView
    a. Load the all_results.vtp file created in step 2
        i. File → Open
        ii. Locate the all_results.vtp file created in step 1
        iii. Click OK





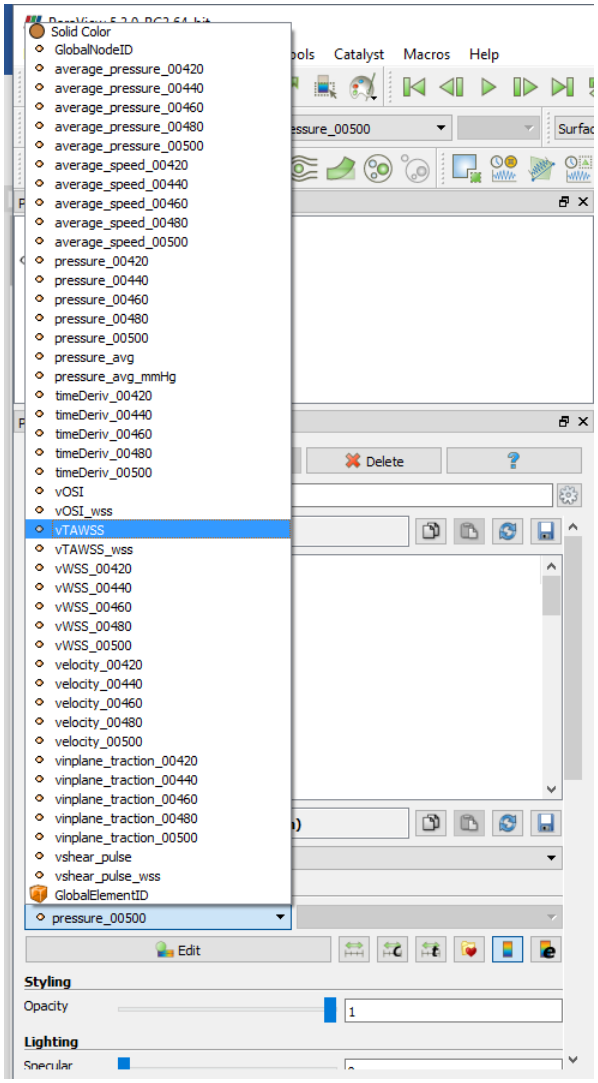
b. all_results.vtp should appear in the Pipeline Browser and multiple properties should appear in the property window
    i. Click Apply



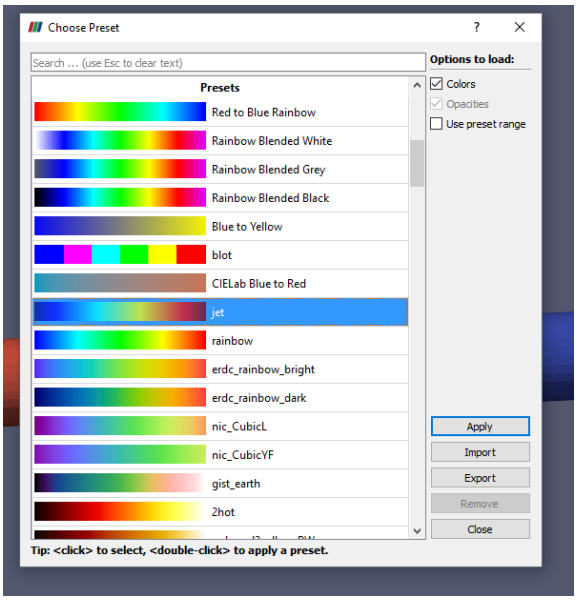
c. The Model should now appear in the layout view

3. IF USER WOULD LIKE TO VIEW TAWSS IN VR

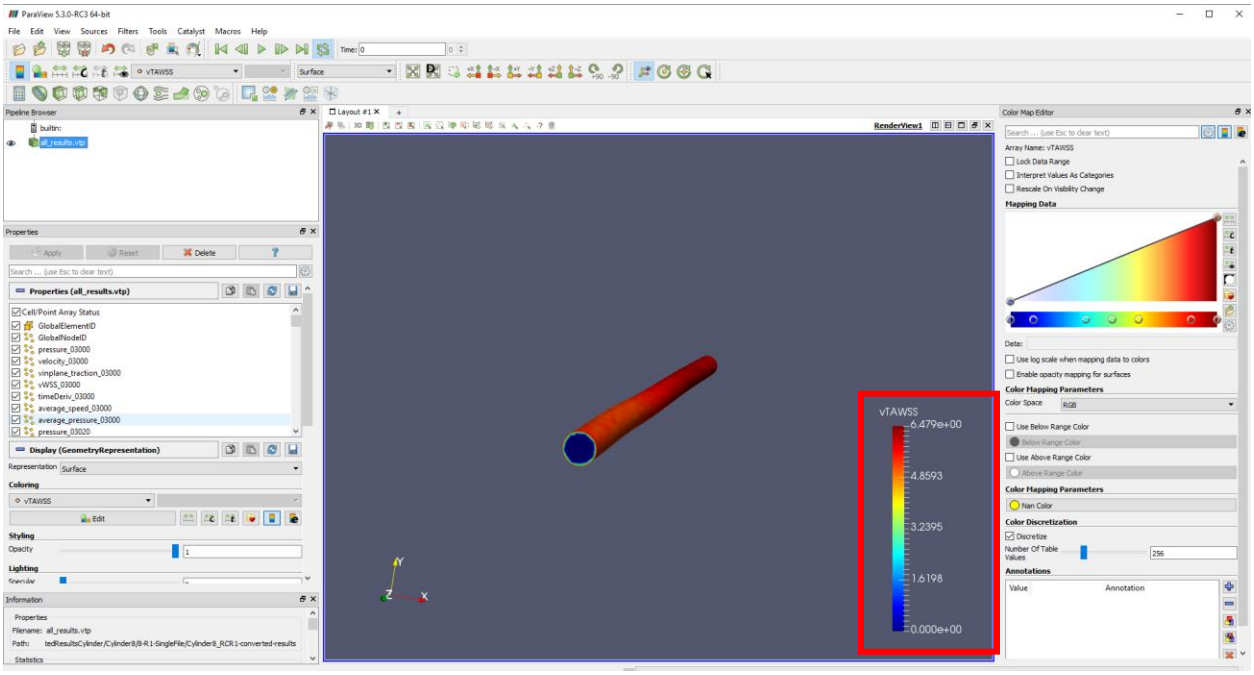
a. In coloring select “vTAWSS”



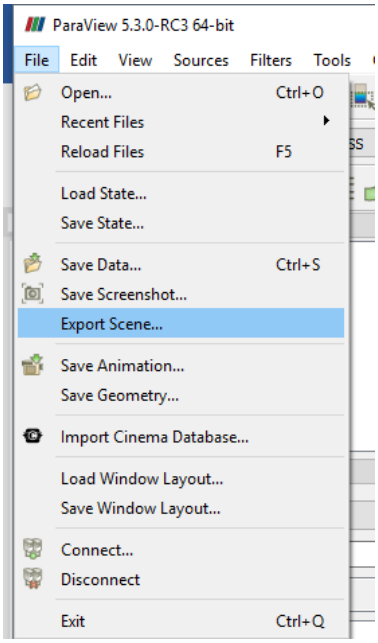
b. Select the “Choose present” button () in the coloring window
   i. A Choose Present Window should appear
   ii. Locate and select “jet” and click “Apply” then click “Close”

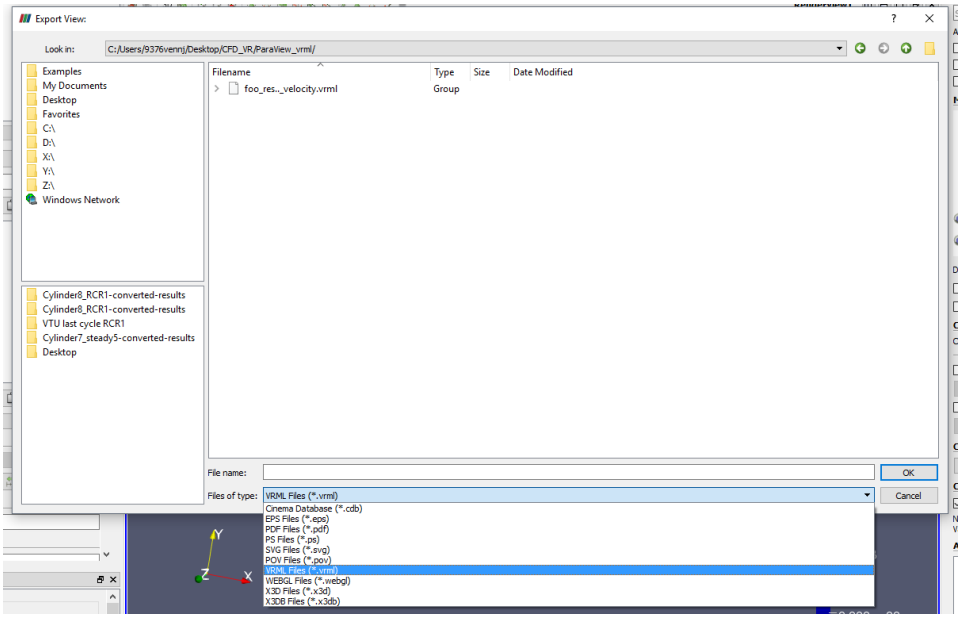


c. Record the min and max values of your scale



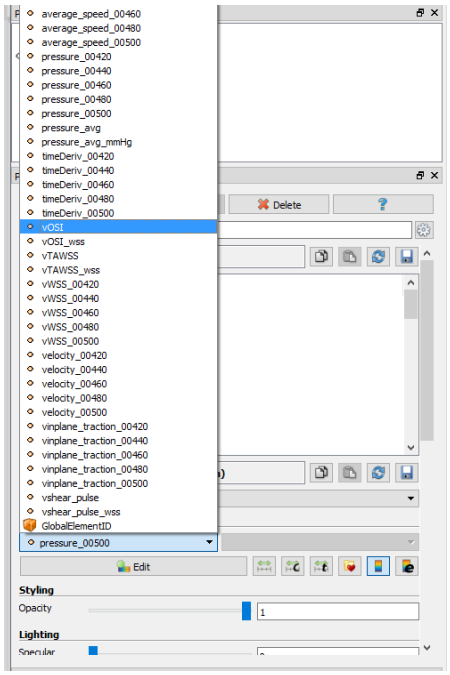
d. Export your scene by clicking File → Export Scene
   i. Locate your CFD_ParaView folder
   ii. In “Files of Type” select VRML Files (*.vrml)
   iii. Save the file as “Mesh_TAWSS”
   iv. Click “OK”



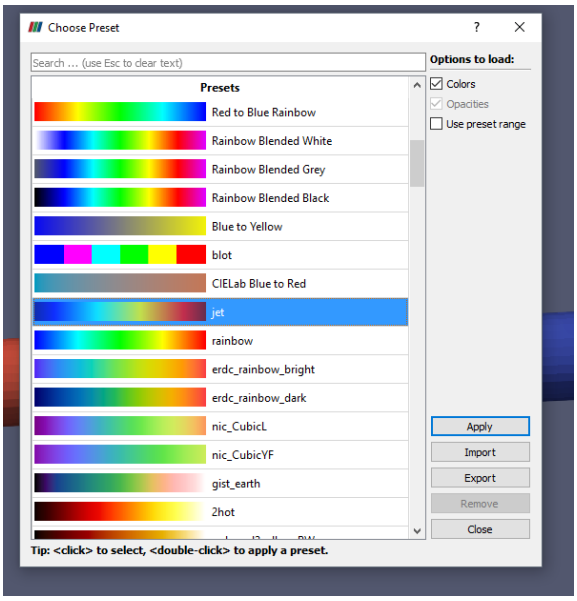


4. IF USER WOULD LIKE TO VIEW OSI IN VR

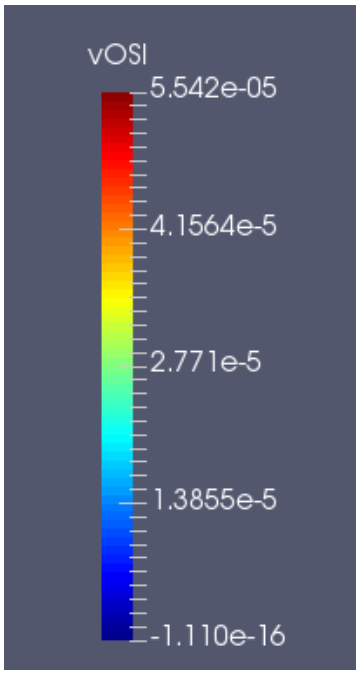
a. In coloring select “vOSI”



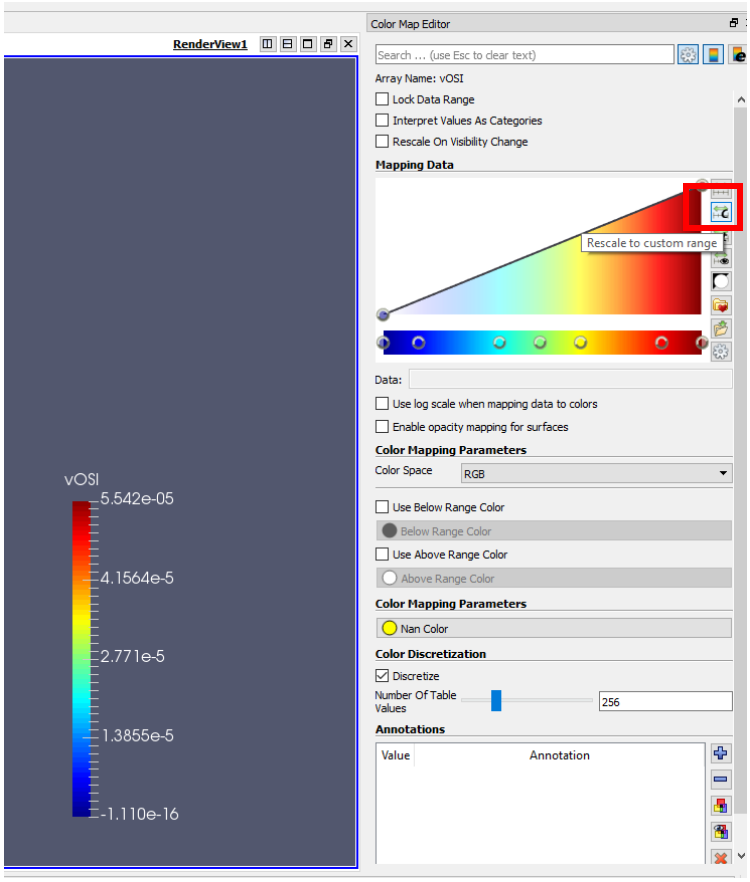
b. Select the “Choose present” button () in the coloring window
    i. A Choose Present Window should appear
    ii. Locate and select “jet” and click “Apply” then click “Close”



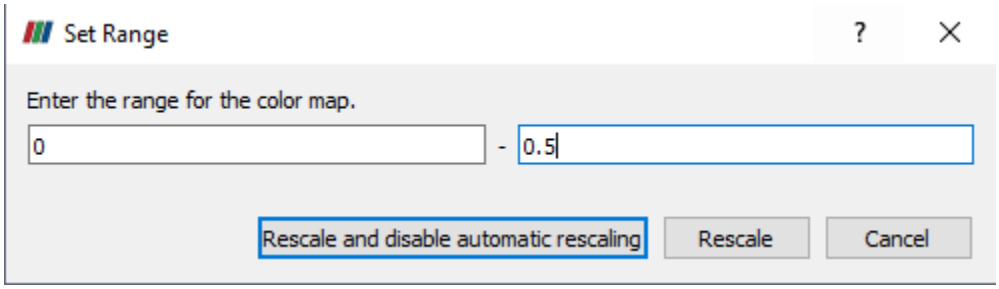
c. IF THE SCALE IS NOT CLOSE TO BEING 0-0.5 if must be adjusted
   i. Example



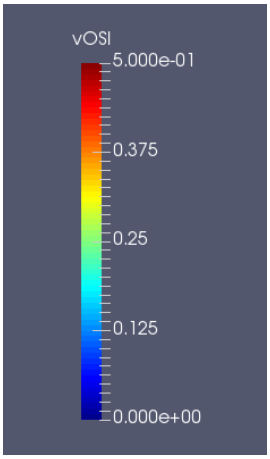
   ii. In the color map editor select “Rescale to custom range” button ()



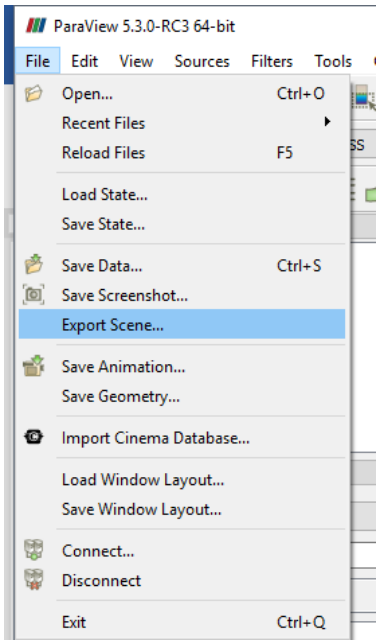
iii. Reset the scale to be 0 and 0.5 then select “Rescale and disable automatic rescaling”

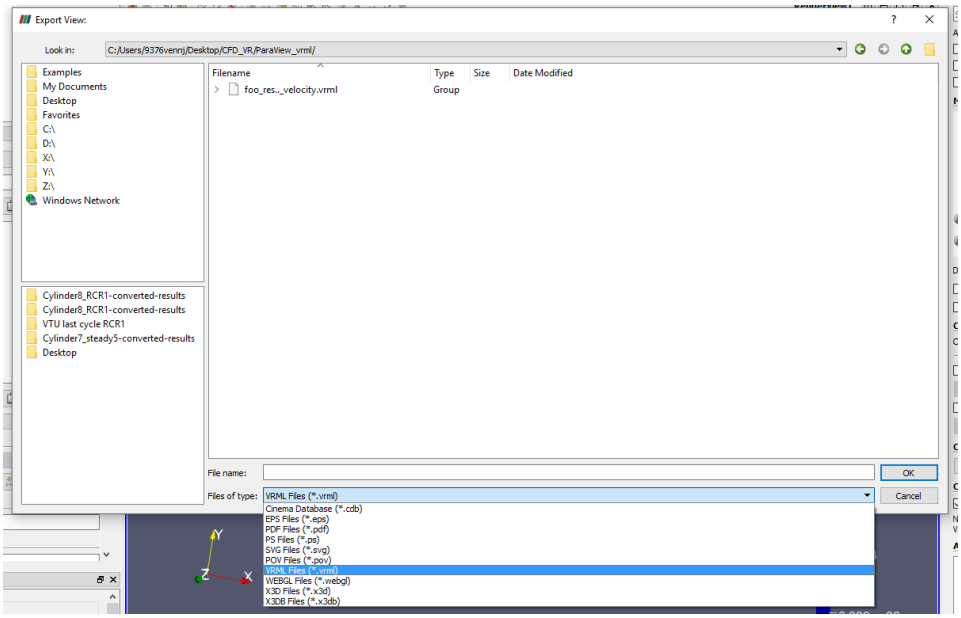


iv. The model should then appear rescaled



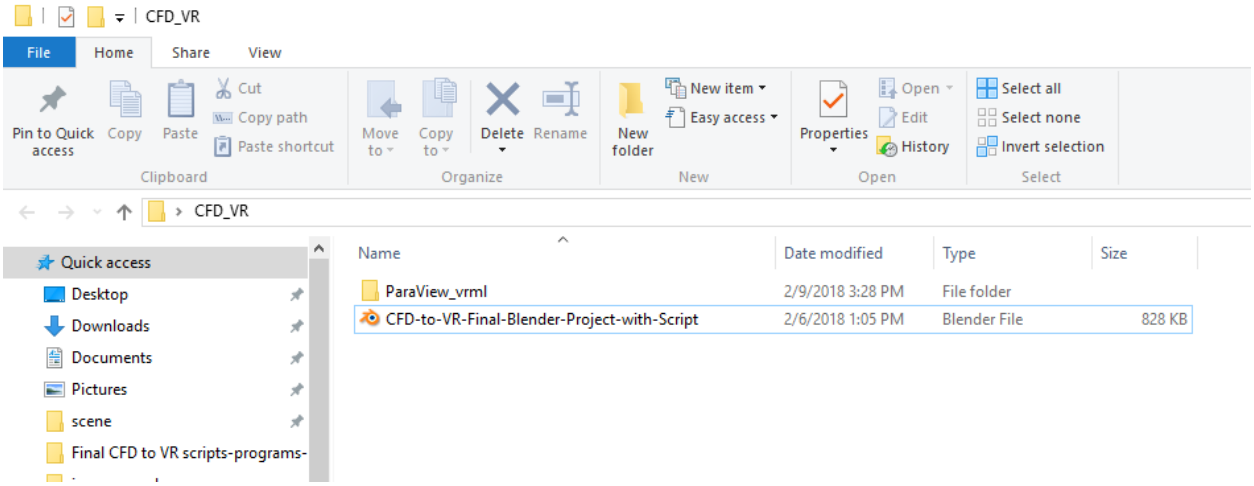
d. Export your scene by clicking File → Export Scene
    i. Locate your CFD_ParaView folder
    ii. In “Files of Type” select VRML Files (*.vrml)
    iii. Save the file as “Mesh_OSI”
    iv. Click “OK”



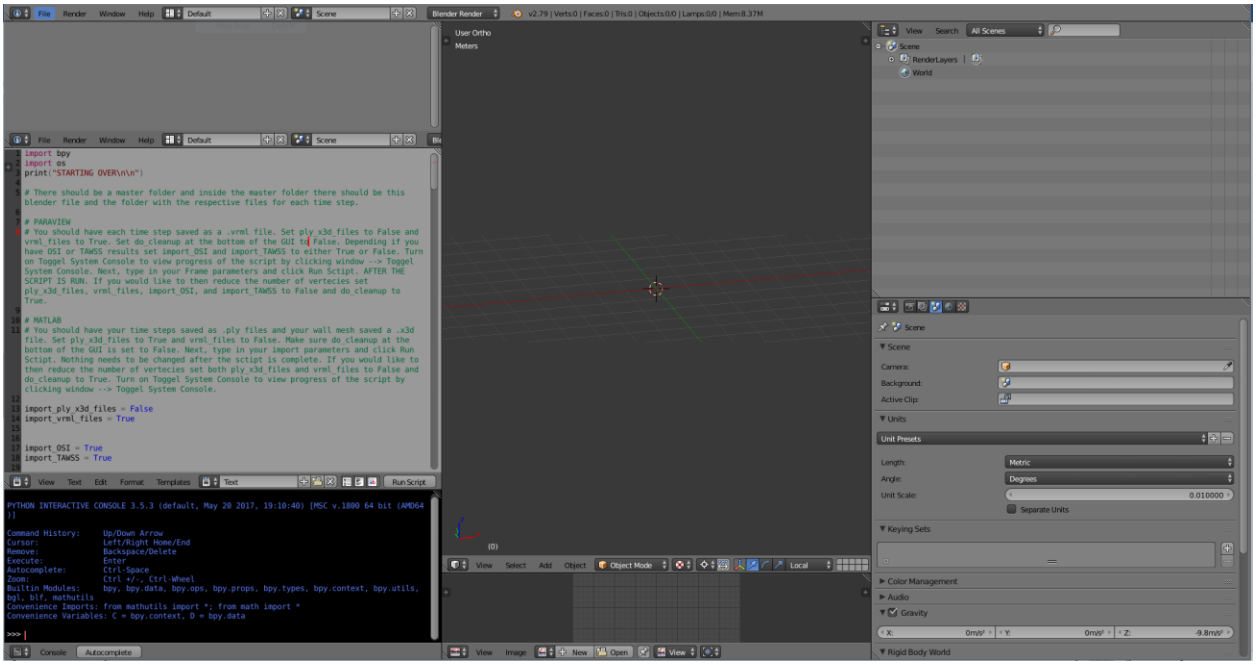

Export View:
Look in:
C:/Users/9376vennj/Desktop/CFD_VR/ParaView_vrml/
Examples
My Documents
Desktop
Favorites
C:\
D:\
X:\
Y:\
Z:\
Windows Network
Cylinder8_RCR1-converted-results
Cylinder8_RCR1-converted-results
VTU last cycle RCR1
Cylinder7_steady5-converted-results
Desktop
Filename
Type
Size
Date Modified
foo_res..._velocity.vrml
Group
File name:
OK
Files of type:
VRML Files (*.vrml)
Cancel
Cinema Database (*.cdb)
EPS Files (*.eps)
PDF Files (*.pdf)
PS Files (*.ps)
SVG Files (*.svg)
POV Files (*.pov)
VRML Files (*.vrml)
WEBGL Files (*.webgl)
X3D Files (*.x3d)
X3DB Files (*.x3db)

3 Reformat CFD results using Blender

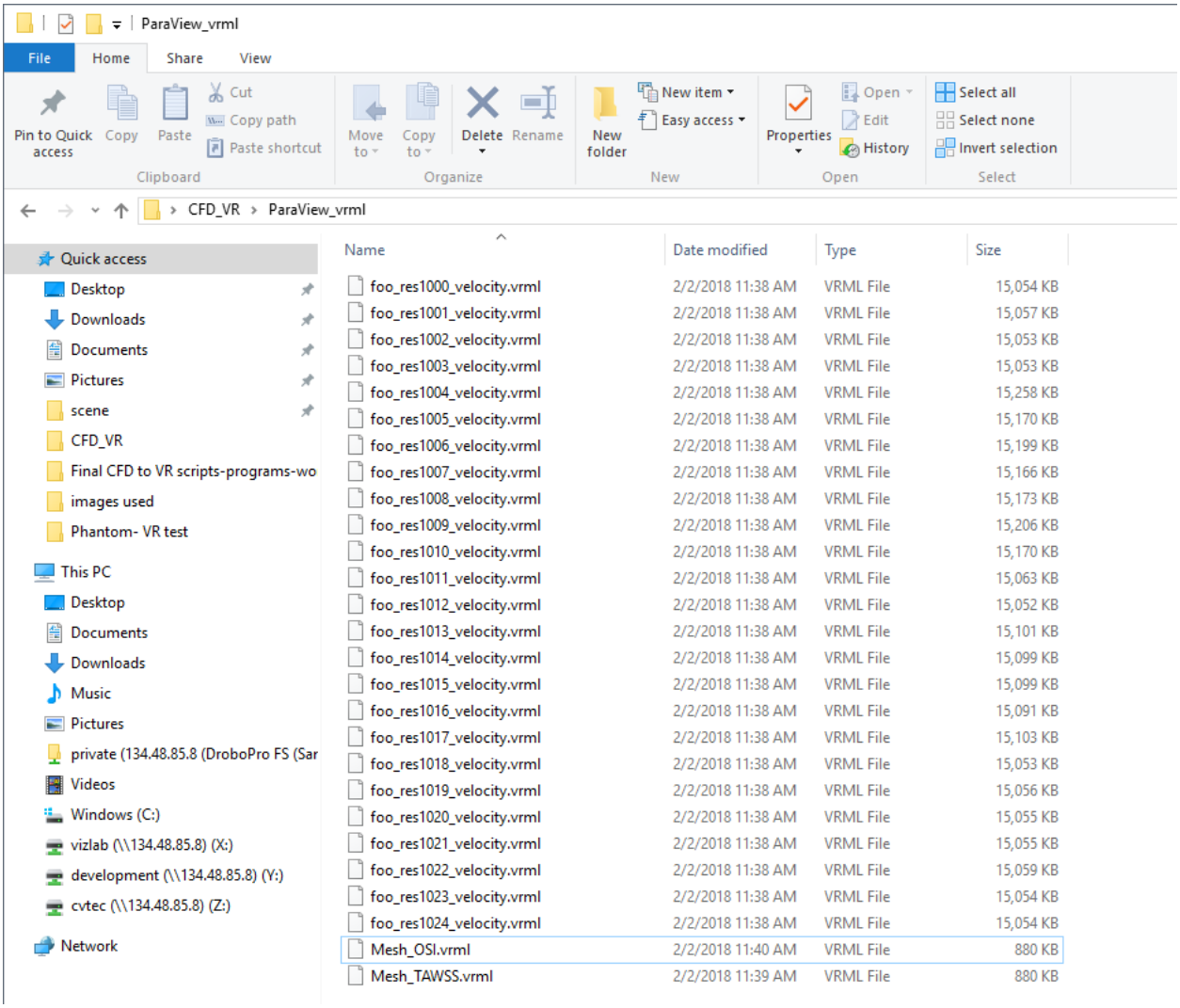
1. Create a new Folder and Call it “CFD_VR”
    a. Drag and drop “CFD-to-VR-Final-Blender-Project-with-Script” blender file into the “CFD_VR” folder
    b. Drag and drop the “ParaView_vrml” folder created in ParaView into the “CFD_VR” folder



2. Open the “CFD-to-VR-Final-Blender-Project-with-Script” blender file. Blender file should look similar to this



3. In the text editor make sure that import_ply_x3d_files is set to false, import_vrml_files is set to true, and depending on if you have OSI or TAWSS set those vales to either true or false
4. In the text editor adjust the start_frame and end_frame to match the first and last files in your ParaView_vrml Folder

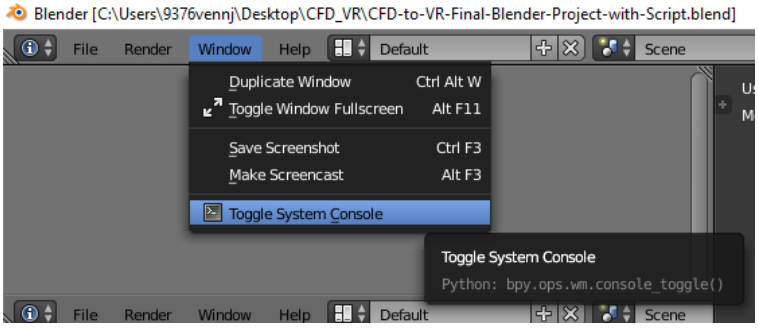


Would type in the following in the text editor

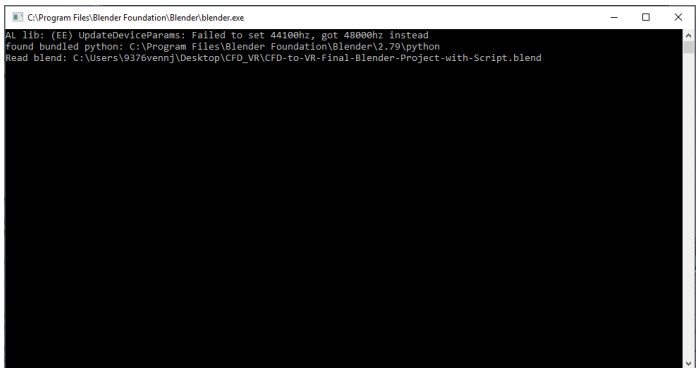
```
import_ply_x3d_files = False
import_vrml_files = True


import_OSI = True
import_TAWSS = True

# Frame parameters- Type in what folder your files are stored in and how each time step
was organized.

folder = "ParaView_vrml" # folder with the wall mesh and streamlines/glyphs data
prefix = "foo_res" # prefix of the streamlines/glyphs data
suffix = "_velocity" # sufix of the streamlines/glyphs data
start_frame = 1000 # start frame or first tme step of the streamlines/glyphs data
end_frame = 1024 # end frame or last timestep of the streamlines/glyphs data
frame_step = 1 # what increment each subsequent time step goes up by

# For example, if you saved your wall mesh and each of your time steps in a
ParaView_vrml folder and your first time step was named foo_res1000_velocity and your
second time step was named foo_res1001_velocity and your last time step was named
foo_res1049_velocity your frame parameters would look like:

#folder = "ParaView_vrml"
#prefix = "foo_res"
#suffix = "_velocity"
#start_frame = 1000
#end_frame = 1049
#frame_step = 1

#Reduce the number of verteceies - will have to change numbers depending on your model-
or could do this using Blender
do_cleanup = False
```

View Text Edit Format Templates Text Run Script

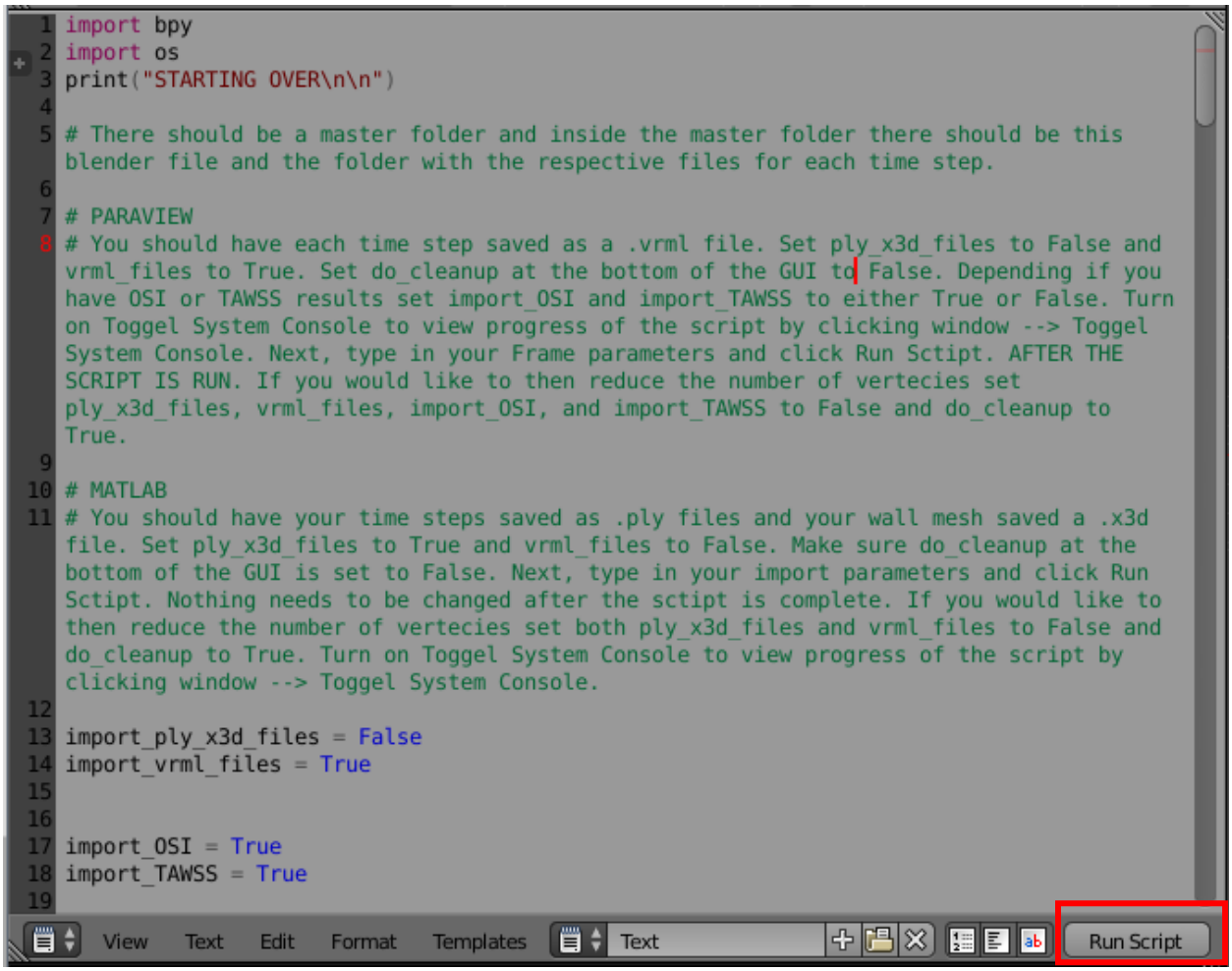
5. Turn on System Counsel by selecting Window → Toggle System Console



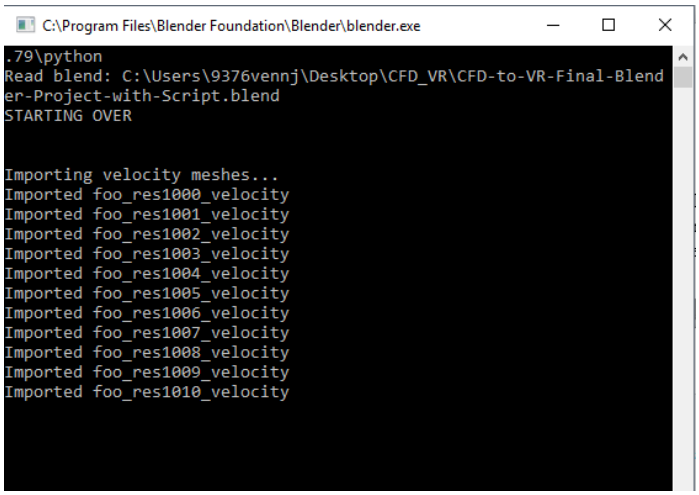
A new window like this should pop up



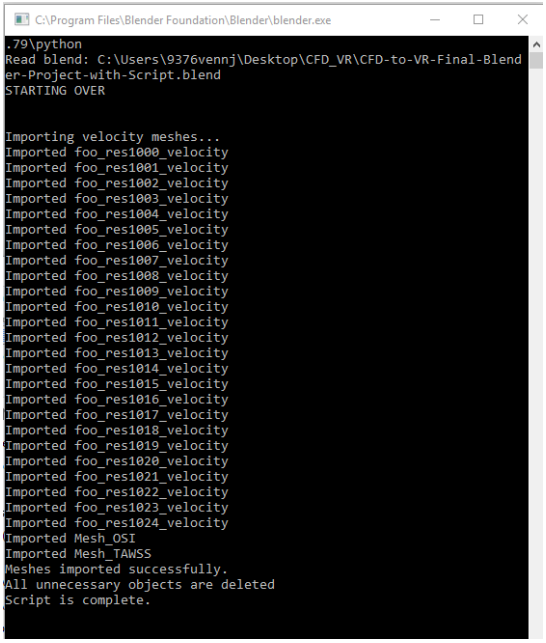
6. In the Text editor window select “Run Script”



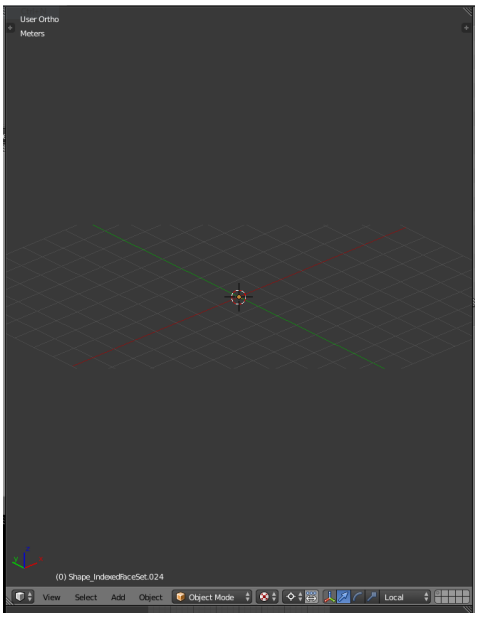
a. While running the System Counsel should keep track of the time steps being loaded



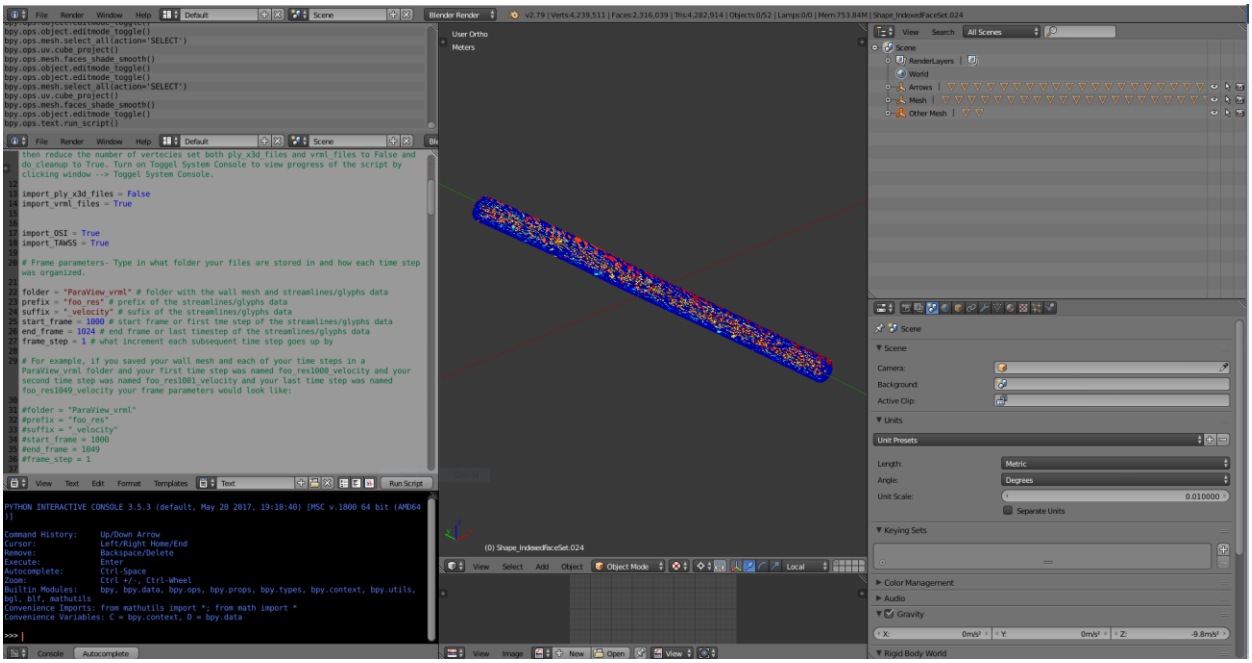
b. After the script is complete, the System Counsel should appear something like



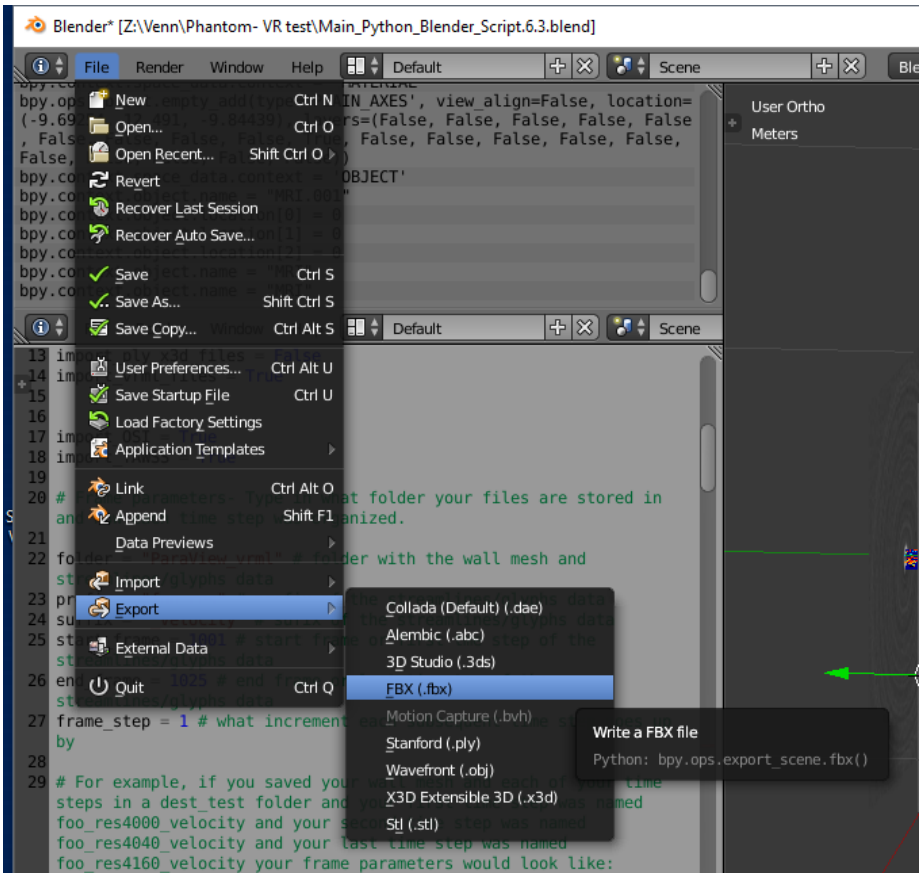
7. After the script if finished Click on the 3D window () and use the mouse middle scroll wheel to zoom (moving in the up direction) in on your model

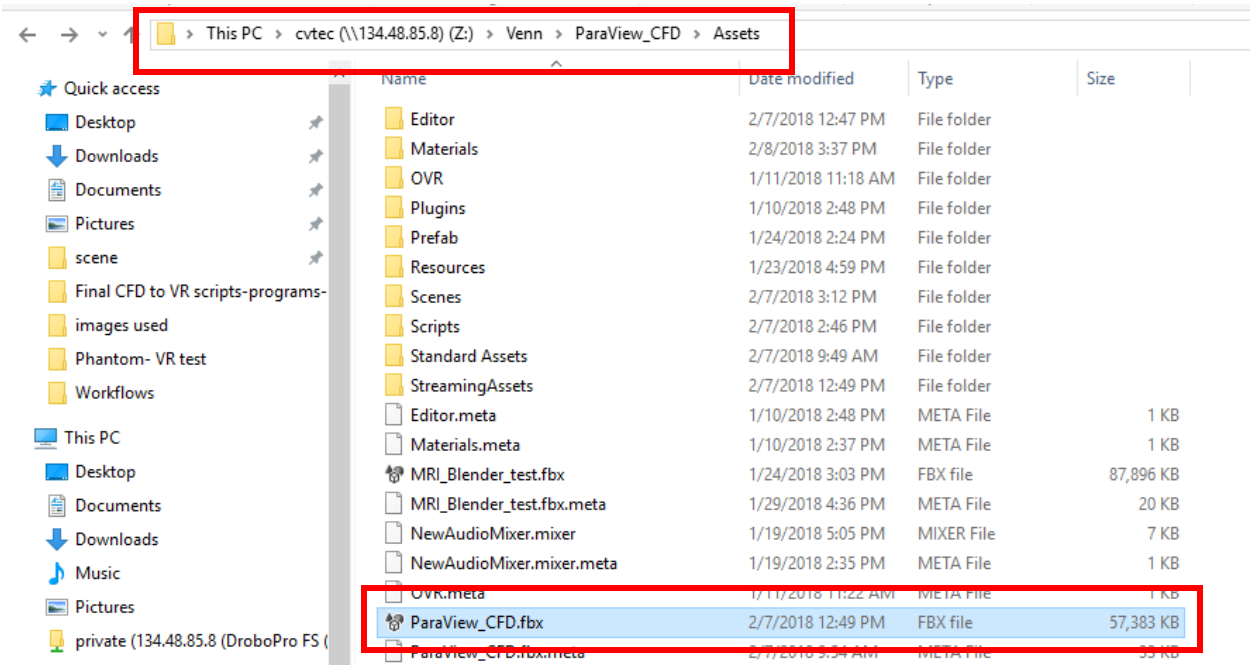


8. After zooming in, your Blender scene should appear something similar to



1) Export your Blender project as an ParaView_CFD.fbx file by going to File → Export → FBX
    a. Replace the FBX file in your Unity/Assets Folder


Blender* [Z:\Venn\Phantom- VR test\Main_Python_Blender_Script.6.3.blend]
File
Render
Window
Help
Default
Scene
New Ctrl N
Open... Ctrl O
Open Recent... Shift Ctrl O
Revert
Recover Last Session
Recover Auto Save...
Save Ctrl S
Save As... Shift Ctrl S
Save Copy... Ctrl Alt S
User Preferences... Ctrl Alt U
Save Startup File Ctrl U
Load Factory Settings
Application Templates
Link Ctrl Alt O
Append Shift F1
Data Previews
Import
Export
External Data
Quit Ctrl Q
Collada (Default) (.dae)
Alembic (.abc)
3D Studio (.3ds)
FBX (.fbx)
Motion Capture (.bvh)
Stanford (.ply)
Wavefront (.obj)
X3D Extensible 3D (.x3d)
Stl (.stl)
Write a FBX file
Python: bpy.ops.export_scene.fbx()
User Ortho
Meters
frame_step = 1 # what increment
by
For example, if you saved your
steps in a dest_test folder and
foo_res4000_velocity and your
foo_res4040_velocity and your last time step was named
foo_res4160_velocity your frame parameters would look like:

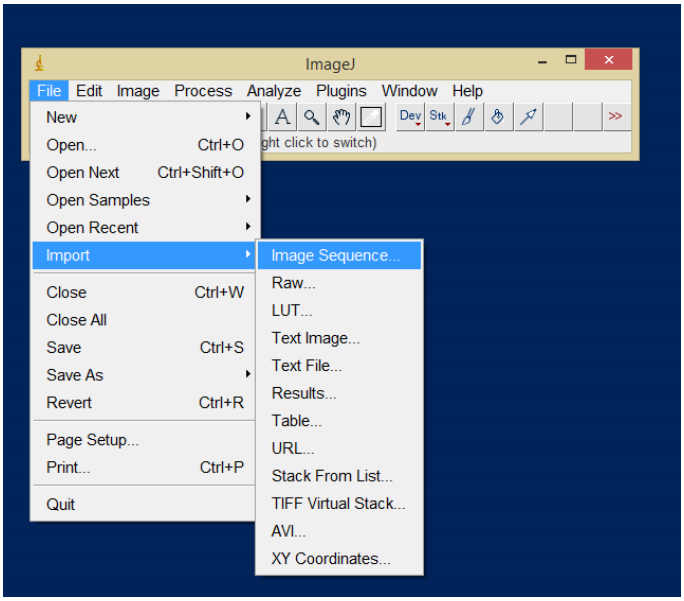

This PC › cvtec (\\134.48.85.8) (Z:) › Venn › ParaView_CFD › Assets
Quick access
Desktop
Downloads
Documents
Pictures
scene
Final CFD to VR scripts-programs-
images used
Phantom- VR test
Workflows
This PC
Desktop
Documents
Downloads
Music
Pictures
private (134.48.85.8 (DroboPro FS (
Name
Date modified
Type
Size
Editor 2/7/2018 12:47 PM File folder
Materials 2/8/2018 3:37 PM File folder
OVR 1/11/2018 11:18 AM File folder
Plugins 1/10/2018 2:48 PM File folder
Prefab 1/24/2018 2:24 PM File folder
Resources 1/23/2018 4:59 PM File folder
Scenes 2/7/2018 3:12 PM File folder
Scripts 2/7/2018 2:46 PM File folder
Standard Assets 2/7/2018 9:49 AM File folder
StreamingAssets 2/7/2018 12:49 PM File folder
Editor.meta 1/10/2018 2:48 PM META File 1 KB
Materials.meta 1/10/2018 2:37 PM META File 1 KB
MRI_Blender_test.fbx 1/24/2018 3:03 PM FBX file 87,896 KB
MRI_Blender_test.fbx.meta 1/29/2018 4:36 PM META File 20 KB
NewAudioMixer.mixer 1/19/2018 5:05 PM MIXER File 7 KB
NewAudioMixer.mixer.meta 1/19/2018 2:35 PM META File 1 KB
ParaView_CFD.fbx 2/7/2018 12:49 PM FBX file 57,383 KB

3.1 Add Volumetric Imaging data using Blender

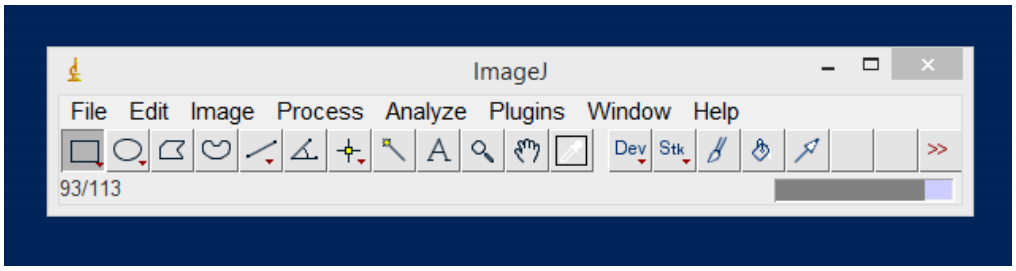
Add MRI/MRA/CT (DICOM) data overlaying model workflow

Image J

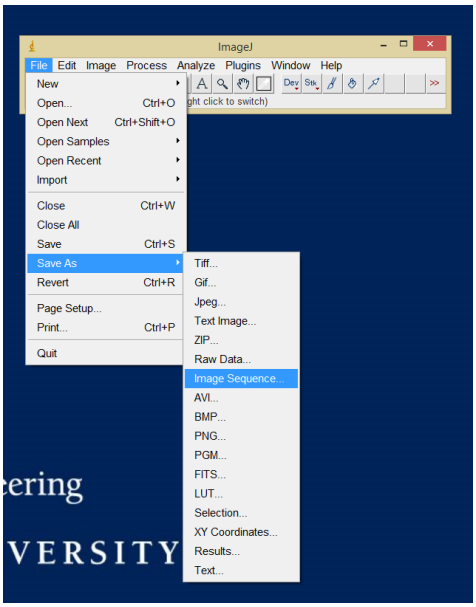
1) Open Image J
2) File → Import → Image sequence
    a. Navigate to the DICOM series

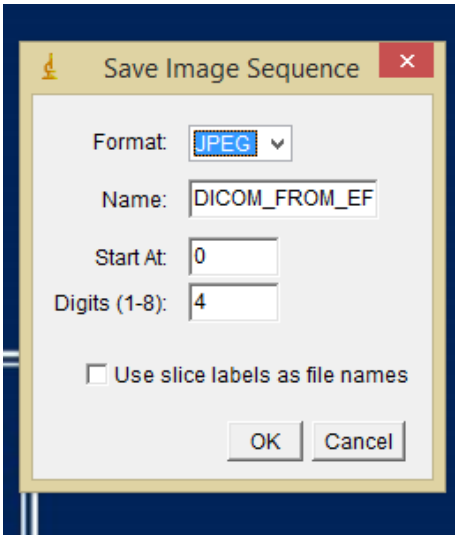


3) Once the DICOM series is located click on the first image and then a Sequence Options screen pops up
    a. Click OK (a loading bar should appear in the lower right hand corner) at the end the DICOM image sequence should appear on the screen



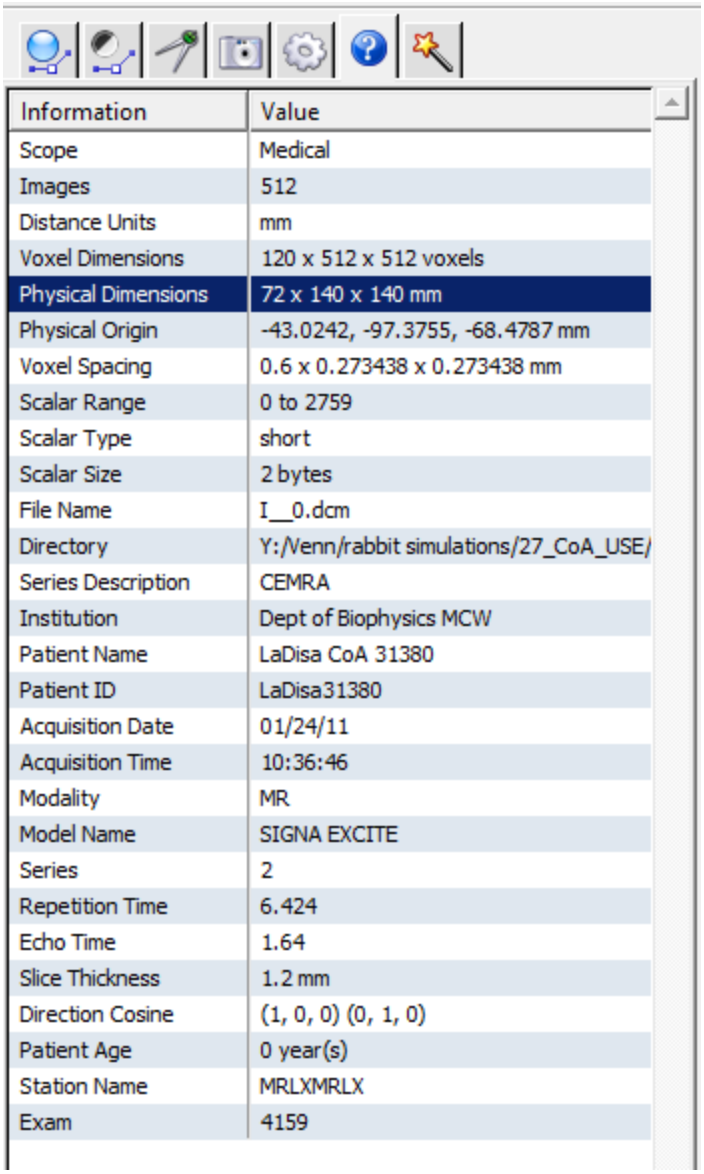
4) Go to file → Save As → Image sequence
    a. For format save the image either as a JPEG or PNG image → ok
    b. Then save and name the image sequence wherever you want to





VolView

1) Open up VolView
2) File → Open File…
    a. Locate your DICOM images you will be uploading and double click on the first image
    b. Click Next >
    c. Click Next >
3) Click on the Info icon () and record the Physical Dimensions and the first number in the Voxel Dimensions (this represents the number of images you have in your image set)
    a. Ex: Images (first number in the Voxel Dimensions) = 120 and Physical Dimensions = 72 x 140 x 140 mm



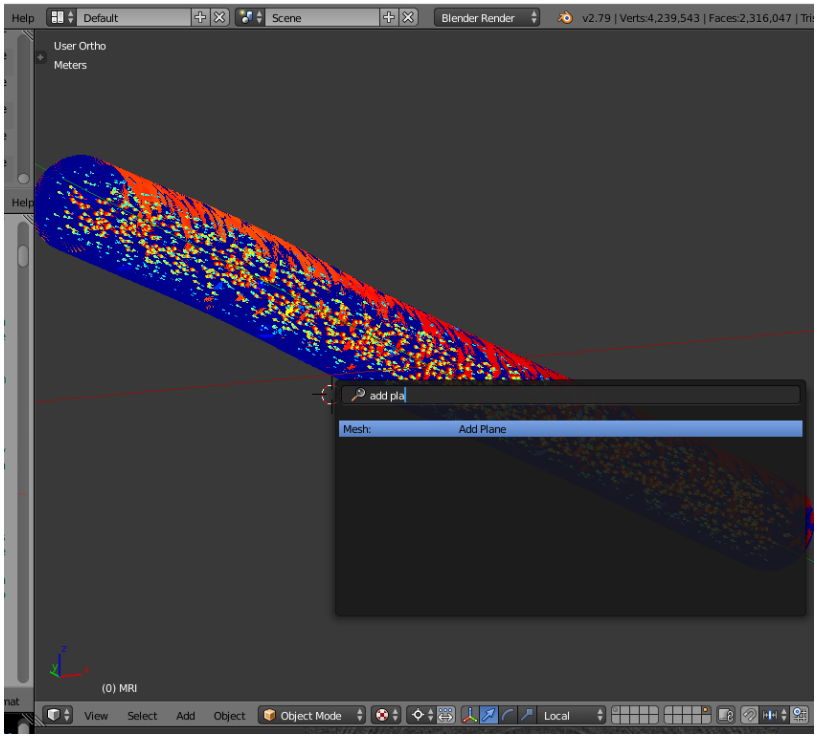
Information	Value
Scope	Medical
Images	512
Distance Units	mm
Voxel Dimensions	120 x 512 x 512 voxels
Physical Dimensions	72 x 140 x 140 mm
Physical Origin	-43.0242, -97.3755, -68.4787 mm
Voxel Spacing	0.6 x 0.273438 x 0.273438 mm
Scalar Range	0 to 2759
Scalar Type	short
Scalar Size	2 bytes
File Name	I__0.dcm
Directory	Y:/Venn/rabbit simulations/27_CoA_USE/
Series Description	CEMRA
Institution	Dept of Biophysics MCW
Patient Name	LaDisa CoA 31380
Patient ID	LaDisa31380
Acquisition Date	01/24/11
Acquisition Time	10:36:46
Modality	MR
Model Name	SIGNA EXCITE
Series	2
Repetition Time	6.424
Echo Time	1.64
Slice Thickness	1.2 mm
Direction Cosine	(1, 0, 0) (0, 1, 0)
Patient Age	0 year(s)
Station Name	MRLXMRLX
Exam	4159

Blender – THE MOST IMPORTANT STEP IS PROPERLY ALIGNING THE FIRST IMAGE THEN IT IS JUST A REPETABLE PROCESS

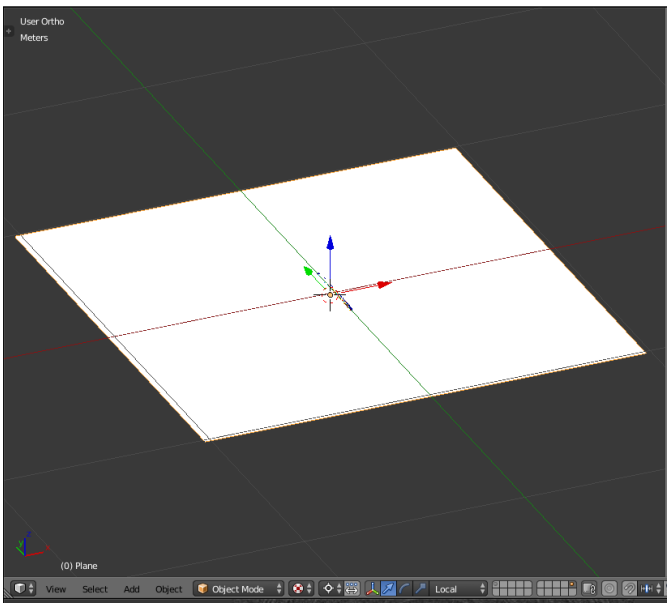
This is assuming that you already ran the Python script therefore, an Arrows, Mesh, and Other mesh parent object were already created, and the subsequent child objects were placed under the respective parent objects

4) While in 3D view () and object mode left click anywhere in the 3D View and press the spacebar on your key board

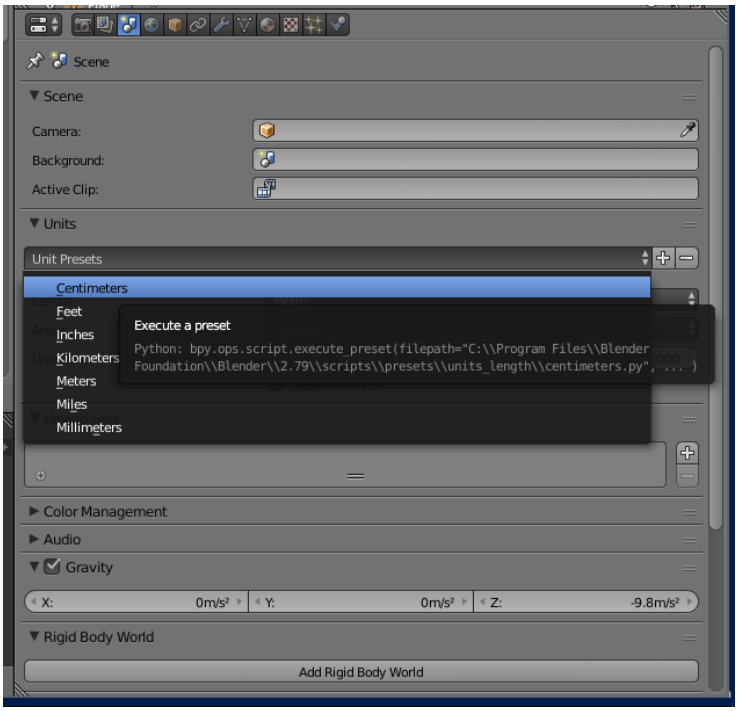
a. Type in Add plane and then click on the Add plane option



b. A white square with an orange outline should now appear

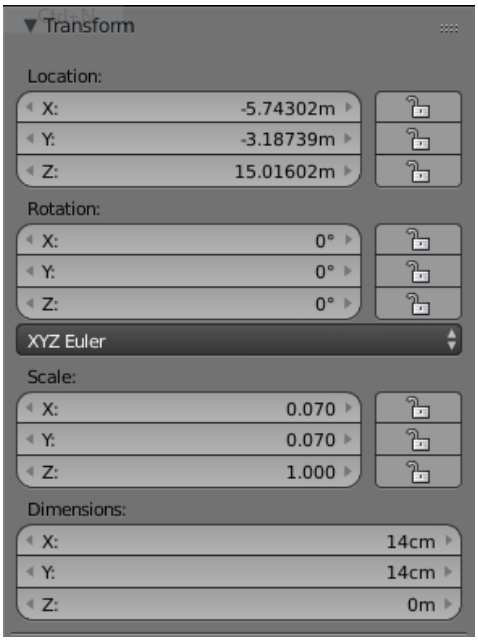


c. Click on the scene icon () and change the Units Presents to Centimeters, Feet, Inches, Kilometers, Meters, Miles or Millimeters



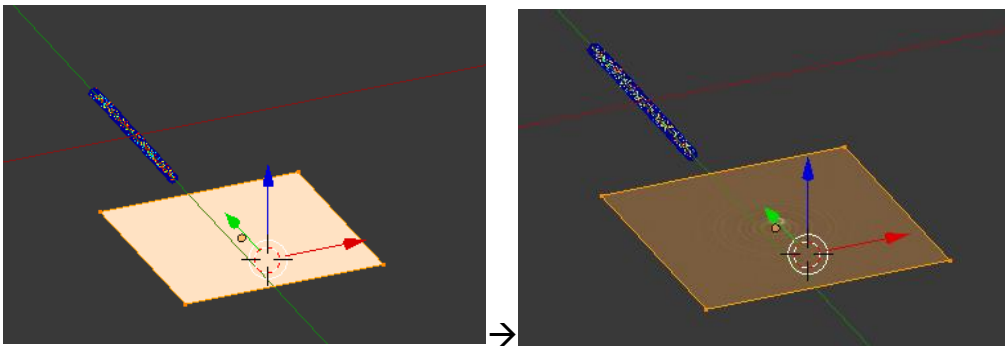
d. In the Transform change the dimensions to what you recorded in the physical dimensions using VolView
    i. To get to the transform, click on the small plus button on the left side of the hierarchy

    ii. Ex: if the physical dimensions were 72 x 140 x 140mm then you would type in 14cm and 14cm in the x and y dimensions



e. Next, scale your plane so it somewhat matches your model
    i. While your plane is still outlined in orange press “G” on your keyboard followed by “S” on your keyboard
        1. Drag your mouse until the image appears to be roughly the same scale as your model
f. While on 3D view switch from Object mode to Edit mode

g. While in UV/Image Editor () and in edit mode click on the Open image icon () and locate the JPEG or PNG image that has a unique slice of your model that would be easy to align



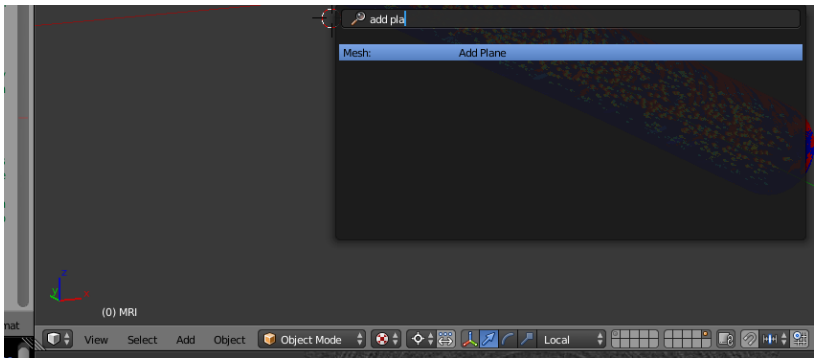
    i. If the image needs to be flipped along the length or width dimensions do the following:

1. Click “S” on the keyboard followed by either clicking or double clicking “X” or “Y” on the keyboard to get the line directly in the plane that the image should be flipped in
2. Then press “-“ followed by “1” on the keyboard
    a. The image should be flipped in the plane you set in part 1

h. In 3D View switch back to object mode

i. While in object mode adjust the scale, rotation and x, y and z planes of your image until the image is properly aligned with your model- **THIS STEP IS VERY IMPORTANT!**
    i. To scale: “g” followed by “s” and then moving your mouse
    ii. To move: “g” followed by “x”, “y” or “z” allows you to grab it and drag it along the desired vertices by then moving your mouse
        1. Note if you double click “x”, “y” or “z” it switches from the global to local orientation
    iii. To rotate: “g” followed by “x”, “y” or “z” followed by “r” allows you to grab it and rotate it along the desired vertices and then moving your mouse
        1. Note if you double click “x”, “y” or “z” it switches from the global to local orientation
j. Once your image is perfectly aligned with your model you can now move to the next step

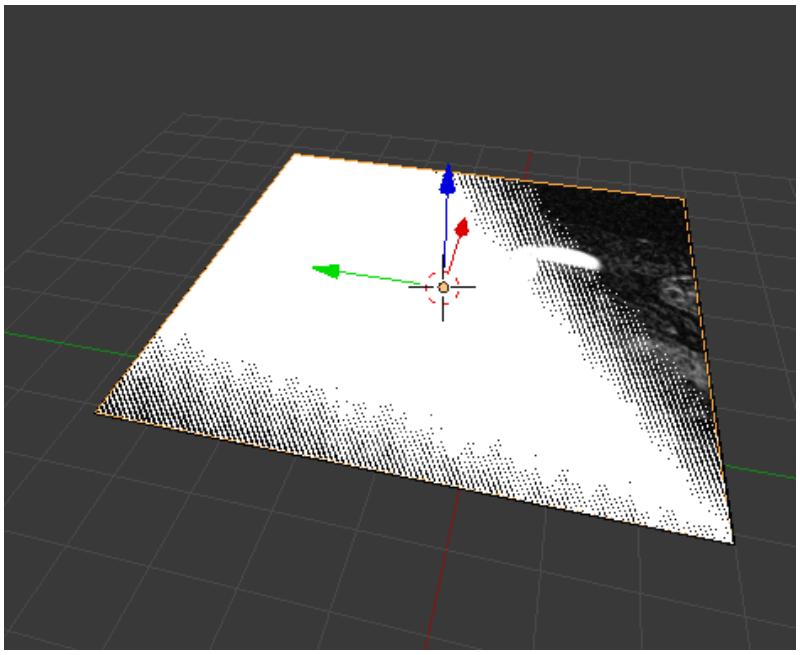
5) While in 3D view () and object mode, left click anywhere in the 3D View and press the spacebar on your key board
    a. Type in Add plane and then click on the Add plane option
    b. A white square with an orange outline should now appear



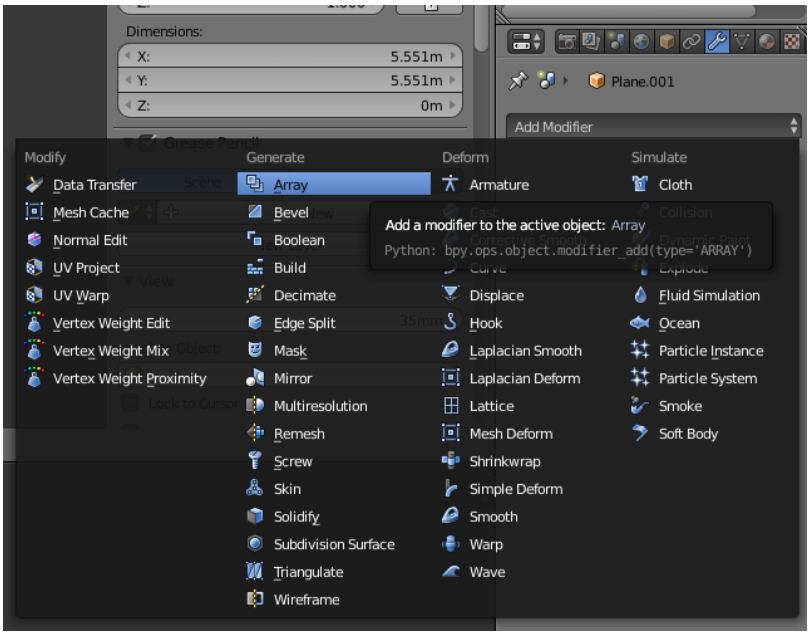
    c. Place the newly created plane directly on the image that has already been aligned by either copying and pasting the Transform Location, Rotation, Scale, and Dimensions from the already aligned image to the newly created plane

(easier) or scaling, rotating, and moving the newly created plane with your key board and mouse (harder)

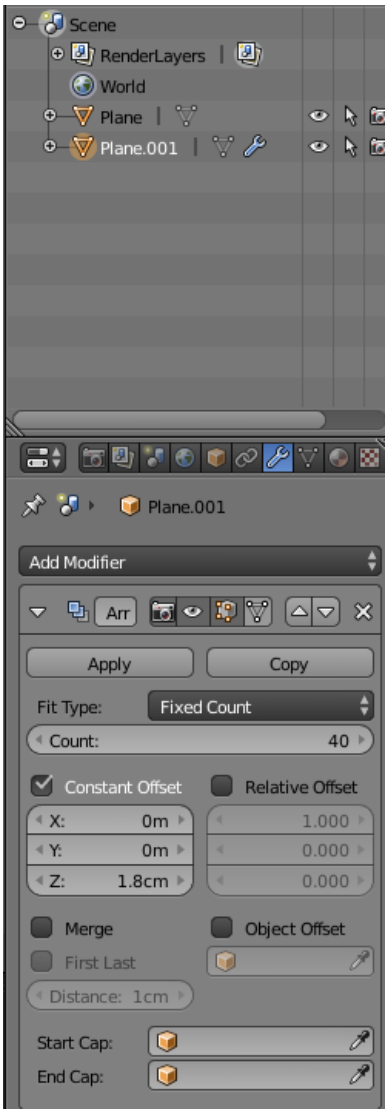
i. To scale: “g” followed by “s” and then moving your mouse
ii. To move: “g” followed by “x”, “y” or “z” allows you to grab it and drag it along the desired vertices by then moving your mouse
    1. Note if you double click “x”, “y” or “z” it switches from the global to local orientation
iii. To rotate: “g” followed by “x”, “y” or “z” followed by “r” allows you to grab it and rotate it along the desired vertices and then moving your mouse
    1. Note if you double click “x”, “y” or “z” it switches from the global to local orientation



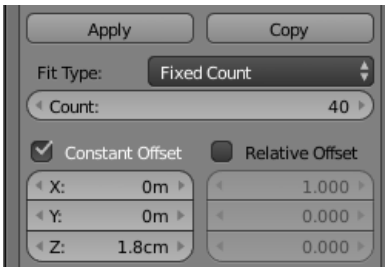
d. Once the newly created plane is directly overlaying the already aligned image Click on the modifiers icon () and click on add Modifier → Array



e. Select the number of “Counts” or number of images you want to upload
    i. This will be important when determining the distance between each image
f. Unselect Relative Offset and Click on Constant Offset
g. Using the number of images you are planning to upload and the physical dimensions depth type in the spacing you would like in the x/y or z direction depending on the orientation your images sequence was taken
    i. Example if you had an image sequence of 120 images, wanting to use all of them and first number in the physical dimensions was 72mm 72mm/119images = 0.605mm/image would be entered in constant offset Z
    ii. But, if you wanted to use every third image in an image sequence of 120 images 0.605mm/image * 3 = 1.815mm would be entered in constant offset Z with your image count now being 40
        1. NOTE: in blender sometimes the metric units get screwed up so you might need to increase or decrease your number by an order of magnitude of 10- use your own judgment if you think they should be tighter together or not



h. Once you type in the spacing you would like each image to go in the x/y or z direction press enter on your key board and a stack of planes should now appear in the 3D view
i. Place the stack of images so the middle is roughly in the middle of your model
    i. Do this by pressing “g” followed by “z”
        1. **MAKE SURE YOU DO NOT MOVE OR DO ANY ROTATING IN THE OTHER DIRECTIONS THE MODEL SHOULD ALREADY BE PERFECTLY ALIGNED**
j. In the Add Modifier interface Press Apply



k. In 3D View switch from object mode to edit mode- the array should now be highlighted in orange

l. Press “P” on your keyboard and select Separate By loose parts
    i. In your outliner there should now be the same number of planes as your Count

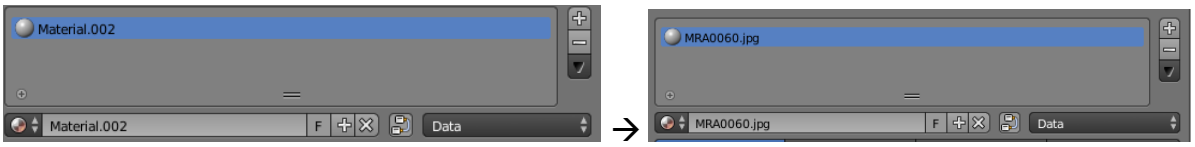
m. In the newly created array locate where the original aligned image is located then find the closest recently added plane
    i. This plane will be used as the starting point to upload the rest of the images

n. Upload the same image as originally aligned image
    i. The image might need to be flipped in the x or y direction or be rotated 90, 180, 270 degrees depending on how blender uploads your image- follow the original instructions how you did this by uploading the first image (steps 4g)

ii. Rename the object () from Plane.xxx to the name of the actual image

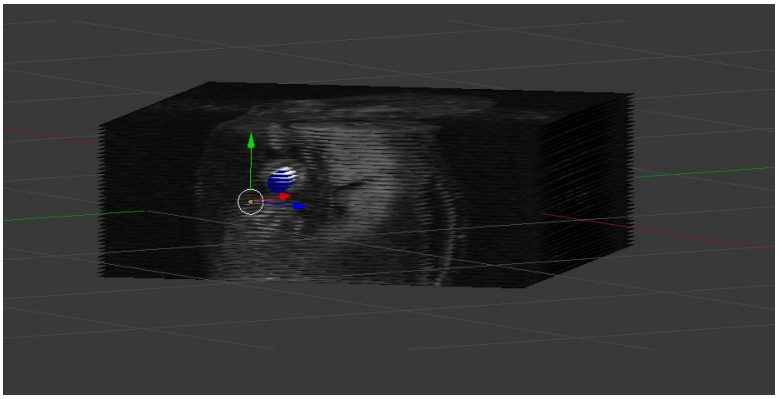
iii. Click on the material icon () and click the + New button

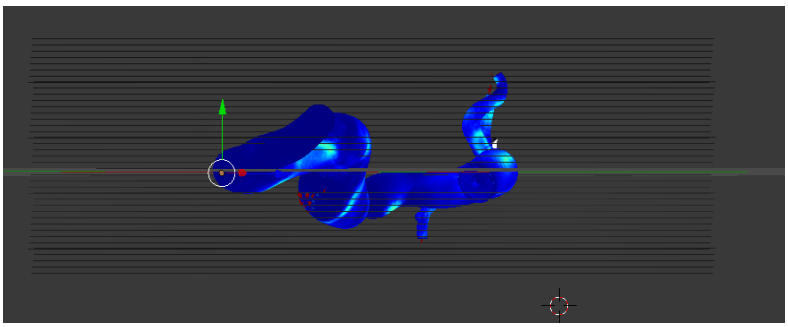
iv. Where it says Material.xxx replace it with the name of the actual image and press enter on your key board

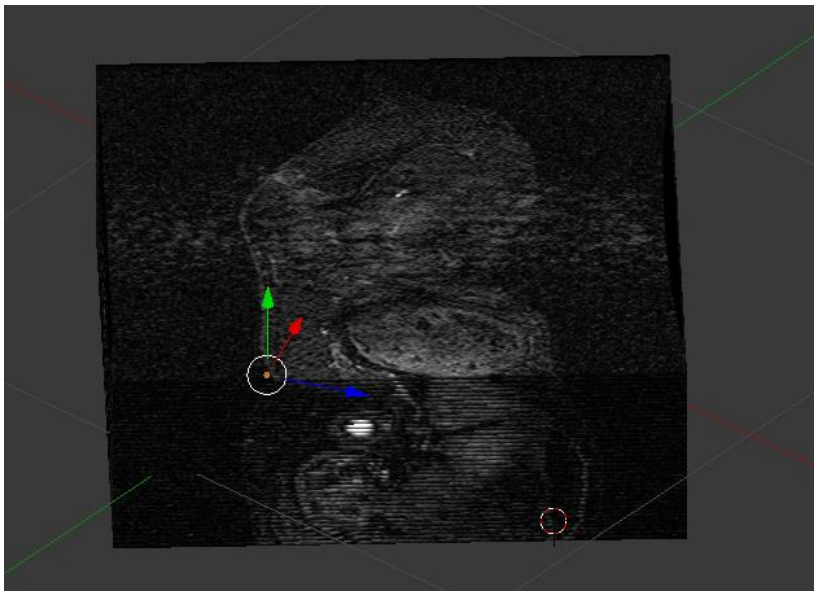


o. Delete the originally aligned image
    i. You basically just made a duplicate of it the step before
p. Select the plane right above the plane that you just uploaded your image and repeat the process of subsequent steps in step n.
    i. Note: if you started at image 60 and you have 120 images and chose your Count as 40 you should select the image 63 as the next image to upload
    ii. Note: it should be the same flipping and rotating process as the first image so it should now be a repetitive process
    iii. Note: it might be easier to hide all of the blank white planes by clicking on the eye icon () (This makes that plane disappear (the eye should turn grey)) until you are uploading that specific planes image
q. At the end you might need to adjust all of the planes if your original alignment was a bit off- this is why it is very important to properly align your first image
    i. Select only the imaging planes (holding ctrl + right click (this should make each plane orange)) and make sure you DO NOT select the Arrows, child arrows object, Other Mesh, child other mesh object, Mesh, and child mesh objects
        1. Another way to do this is use the eye icon to only turn on the image slices then in 3D View and in object mode press “b” on your key board and drag your mouse until each plane is outlined in orange then press “g” followed by the direction you want the image to move or rotate
    ii. Next turn on (by pressing the eye icon) one of the Mesh child game objects so you have something to reference too
    iii. While in object mode Adjust the scale and x, y and z planes of your image stack until the image stack is properly aligned with your model

1. To scale: “g” followed by “s” and then moving your mouse
2. To move: “g” followed by “x”, “y” or “z” allows you to grab it and drag it along the desired vertices by then moving your mouse
    a. Note if you double click “x”, “y” or “z” it switches from the global to local orientation
3. To rotate: “g” followed by “x”, “y” or “z” followed by “r” allows you to grab it and rotate it along the desired vertices and then moving your mouse
    a. Note if you double click “x”, “y” or “z” it switches from the global to local orientation
4. NOTE: these adjustments should be very-very-very small if you properly aligned the original image
5. After alignment, the blender project should look something to the images below



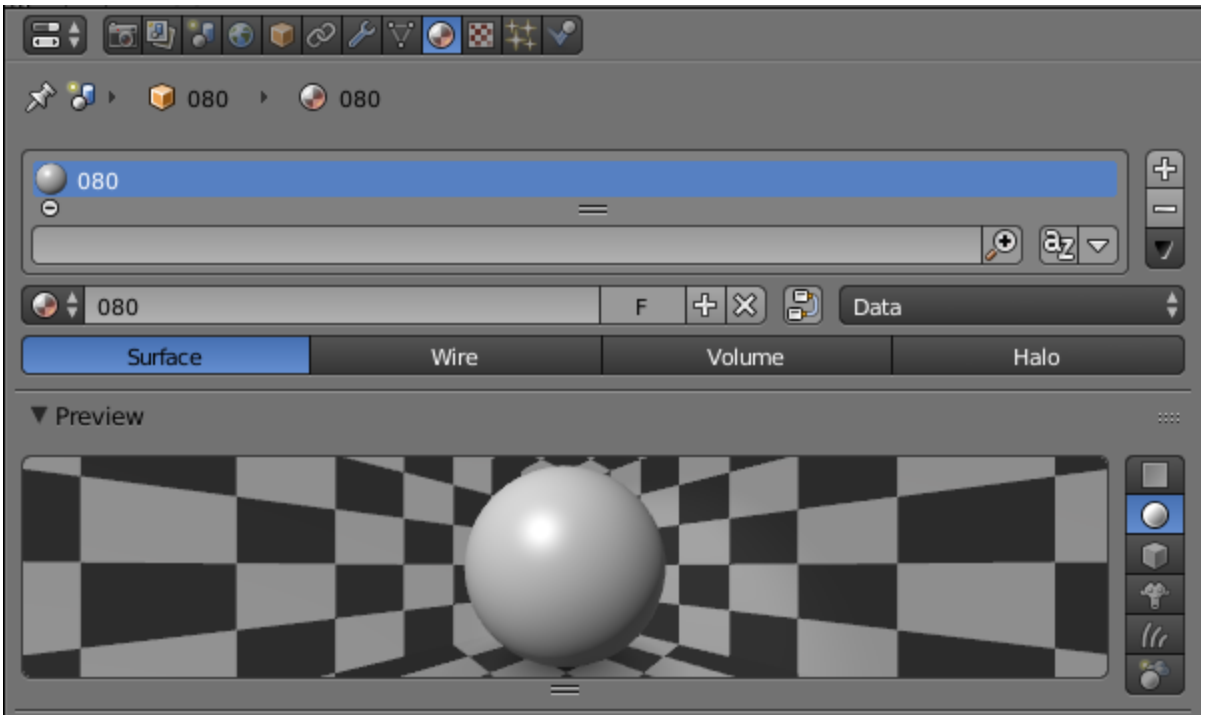




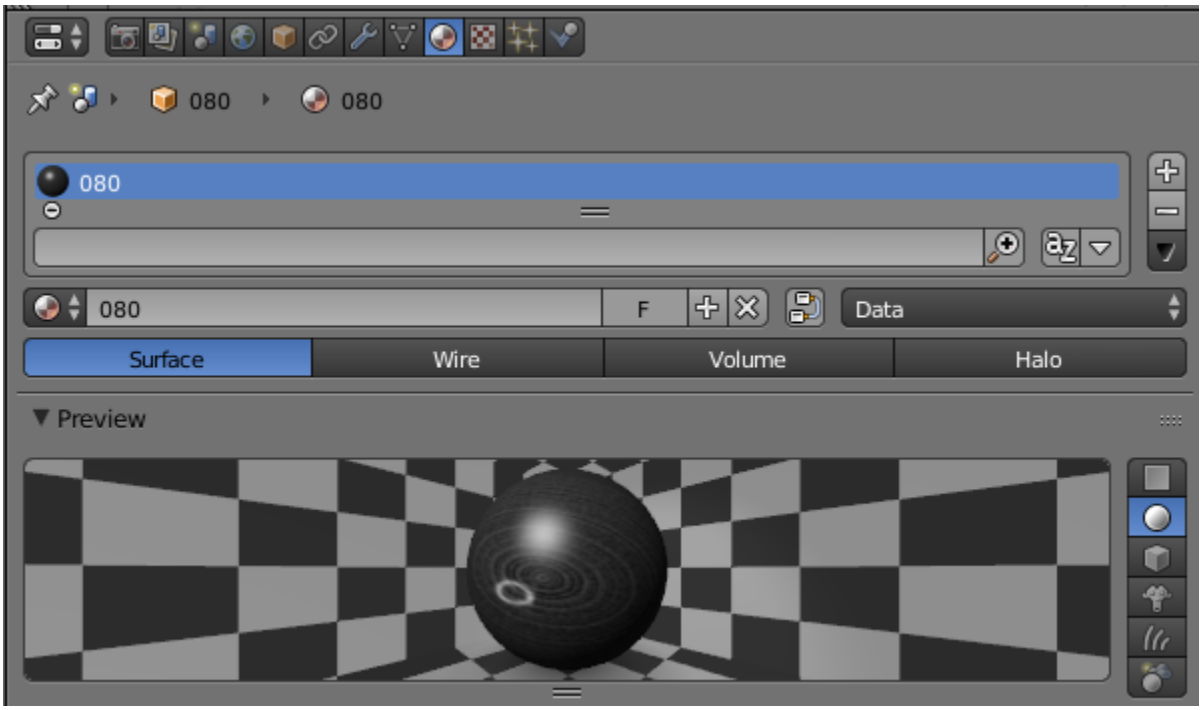
6) Once each plane is properly aligned Texture needs to be added to each plane
    a. Click on your first plane and click on the Texture Plane()
    b. Click on the New +

    c. Click on the Open Image and locate the Image that was selected in part 5.P

    d. When you go to the Material Tab () the sphere should of changed from an all gray sphere into a sphere with your image overlaid on it

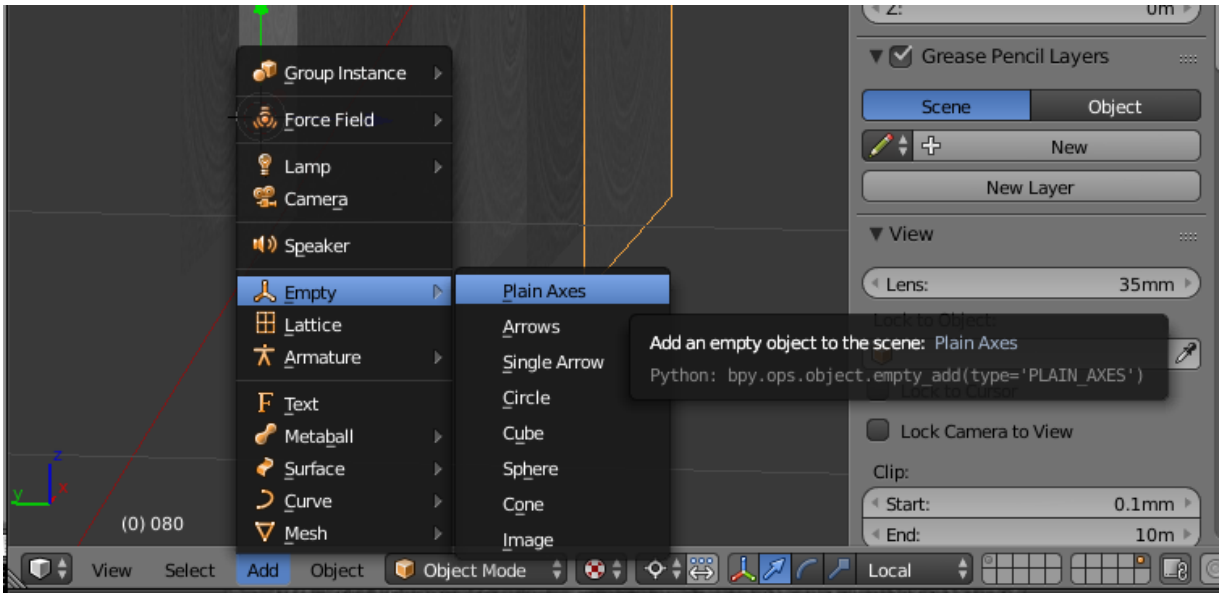


After completing step 6.d



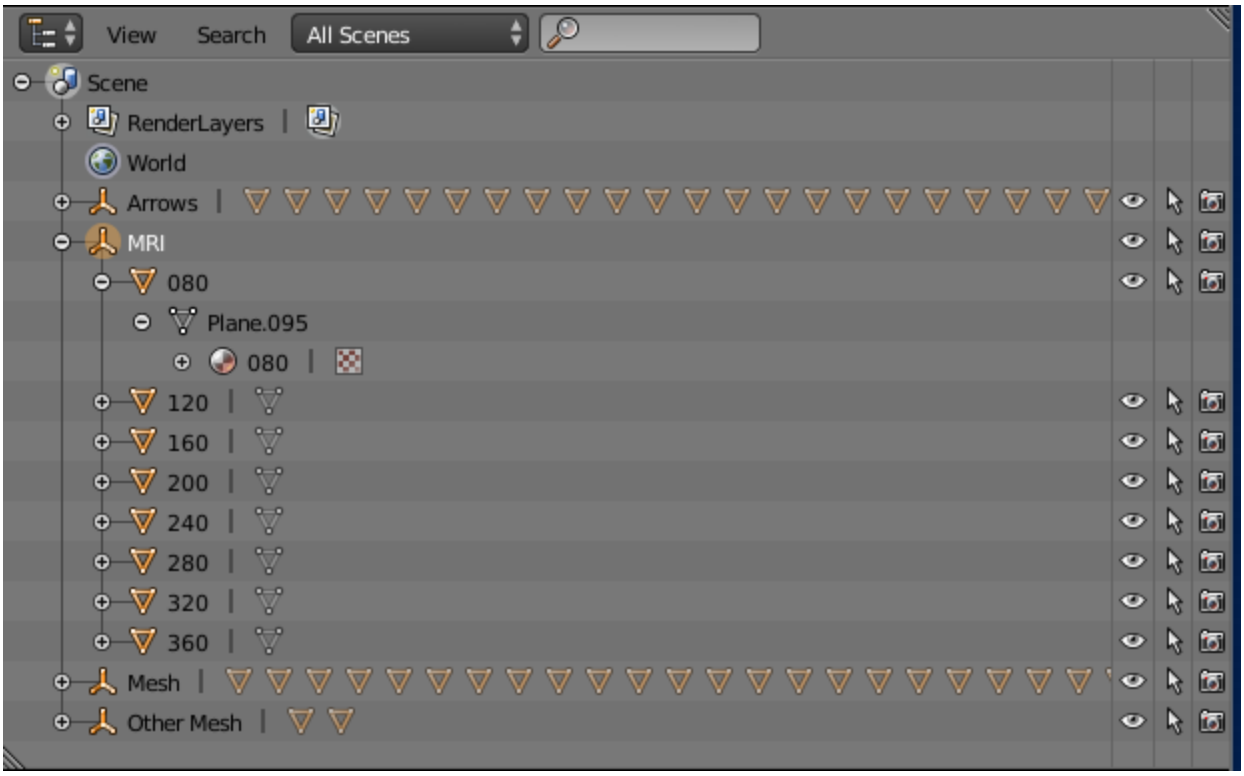
e. Repeat step 6 for the remainder of your plane slices

7) Create a parent object Clicking Add→ Empty → Plain Axes in the 3D view window

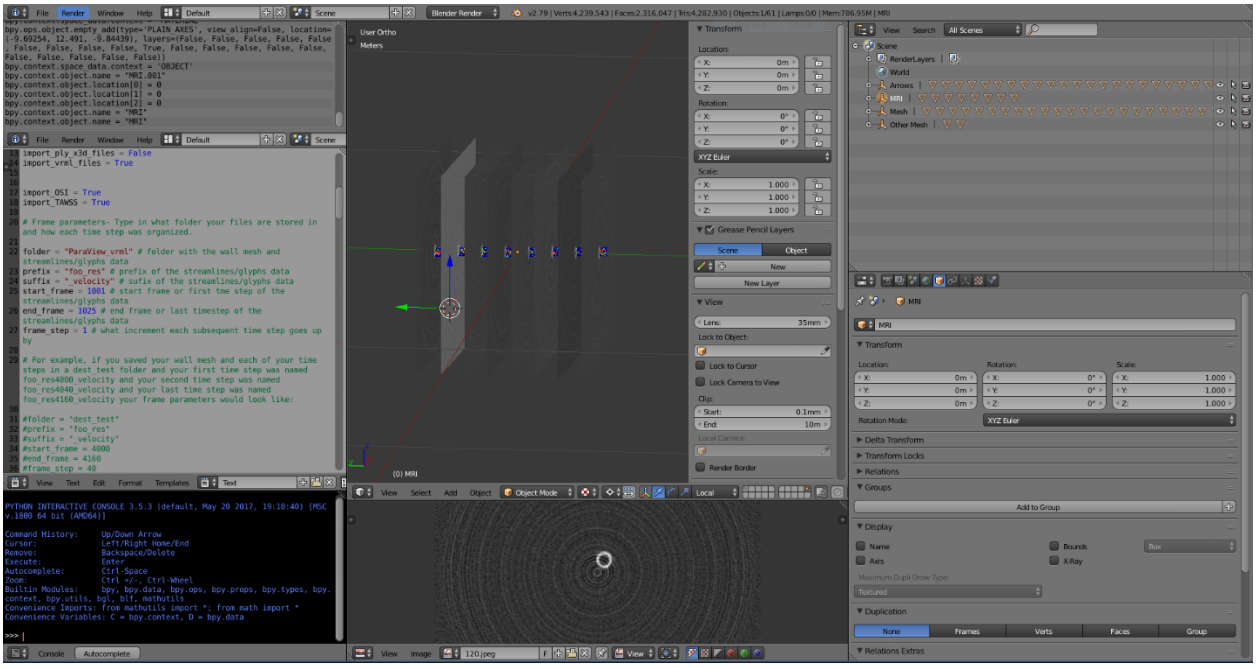


a. On the newly created Empty object click on the Object tab ( ) and change the name from “Empty” to “MRI” and change the location to be X: 0, Y:0, Z:0

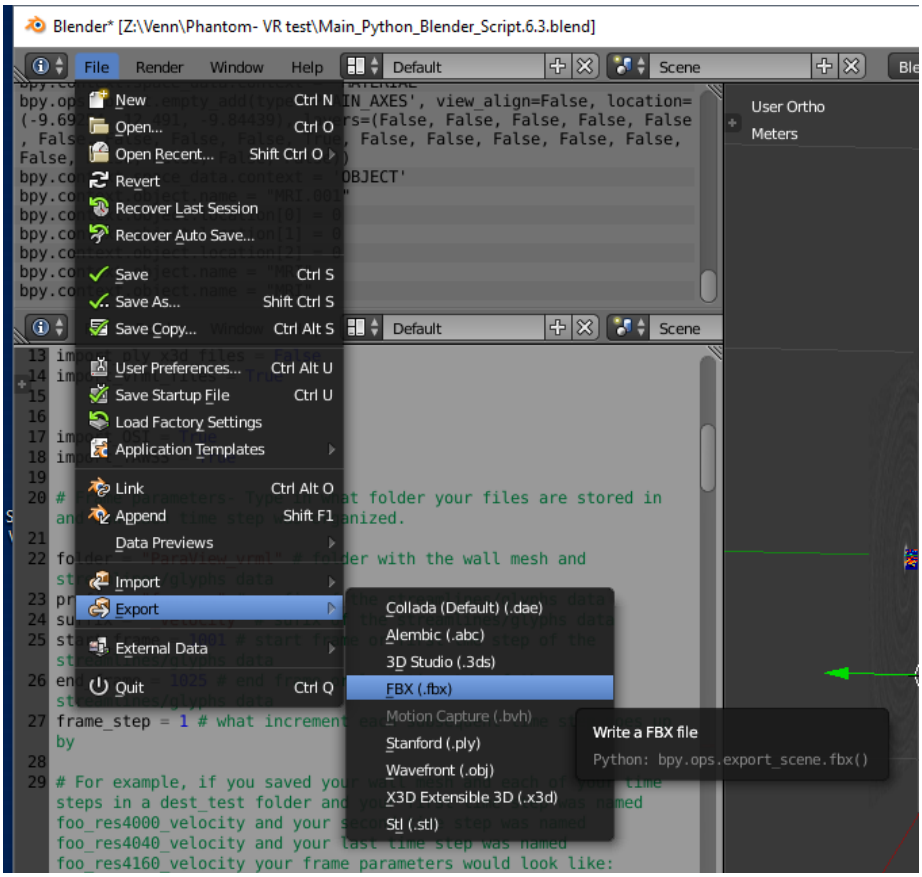
b. Drag and drop each plane with its subsequent Material and Texture into the MRI empty game object, with each plane now considered child objects. The Order should appear something like

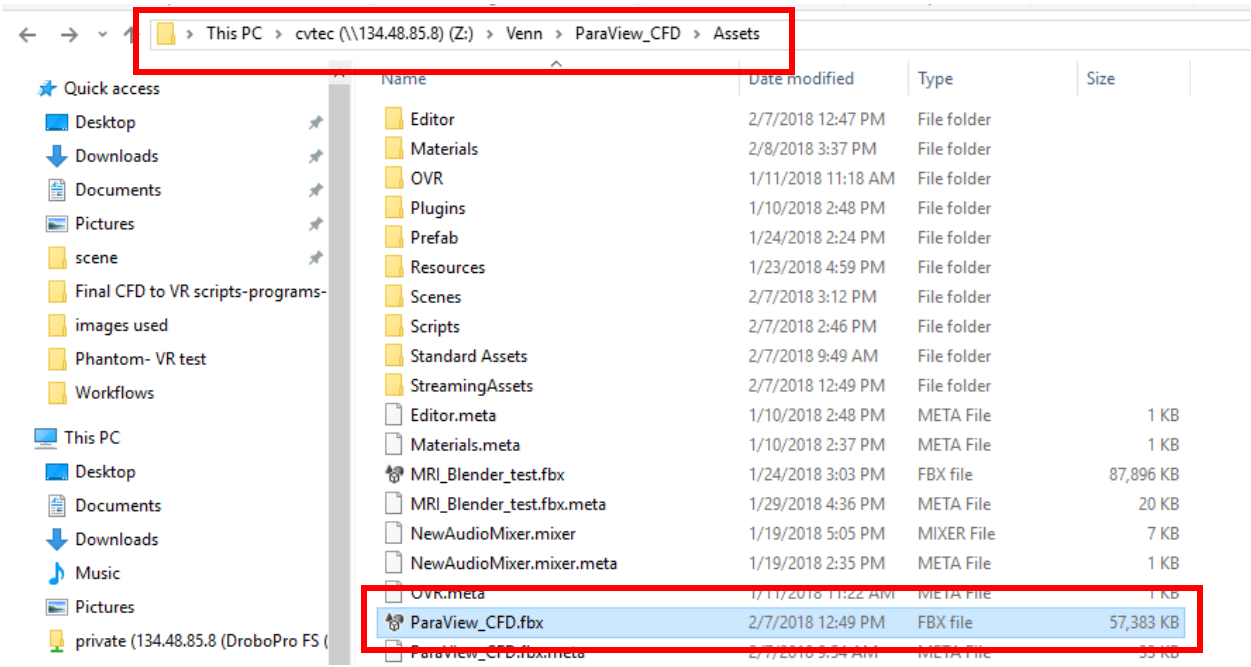


8) Your final project should look something similar to this

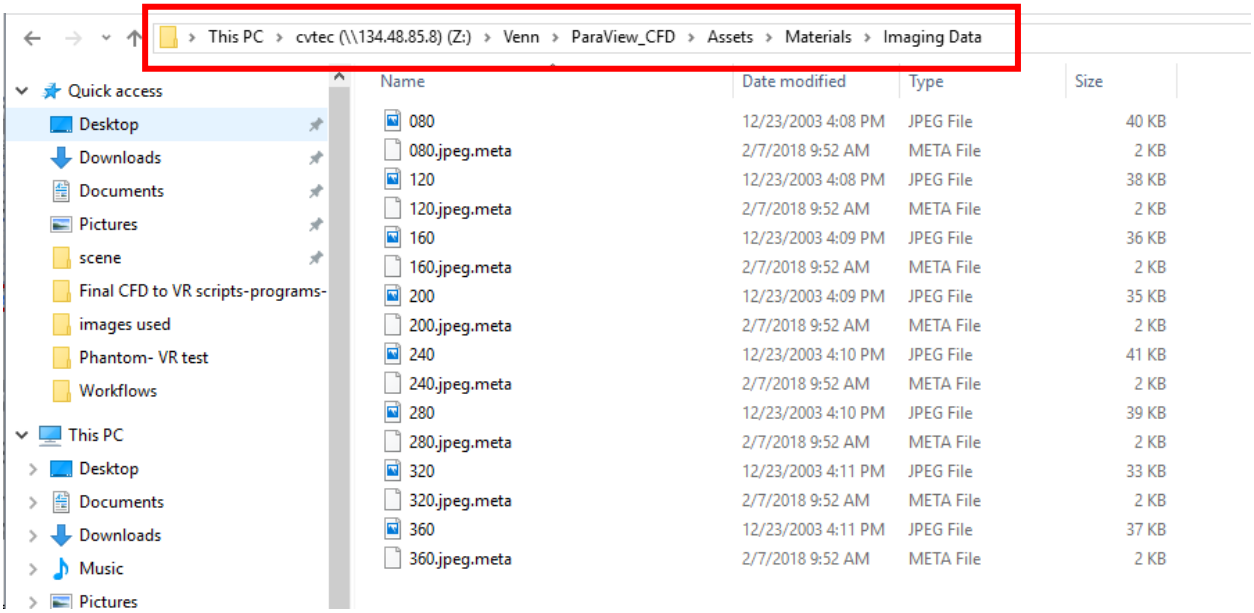


9) Export your Blender project as an ParaView_CFD.fbx file by going to File → Export → FBX
    a. Replace the FBX file in your Unity/Assets Folder


Blender* [Z:\Venn\Phantom- VR test\Main_Python_Blender_Script.6.3.blend]
File
Render
Window
Help
Default
Scene
New Ctrl N
Open... Ctrl O
Open Recent... Shift Ctrl O
Revert
Recover Last Session
Recover Auto Save...
Save Ctrl S
Save As... Shift Ctrl S
Save Copy... Ctrl Alt S
User Preferences... Ctrl Alt U
Save Startup File Ctrl U
Load Factory Settings
Application Templates
Link Ctrl Alt O
Append Shift F1
Data Previews
Import
Export
External Data
Quit Ctrl Q
Collada (Default) (.dae)
Alembic (.abc)
3D Studio (.3ds)
FBX (.fbx)
Motion Capture (.bvh)
Stanford (.ply)
Wavefront (.obj)
X3D Extensible 3D (.x3d)
Stl (.stl)
Write a FBX file
Python: bpy.ops.export_scene.fbx()
User Ortho
Meters
frame_step = 1 # what increment
by
For example, if you saved your
steps in a dest_test folder and
foo_res4000_velocity and your
foo_res4040_velocity and your last time step was named
foo_res4160_velocity your frame parameters would look like:

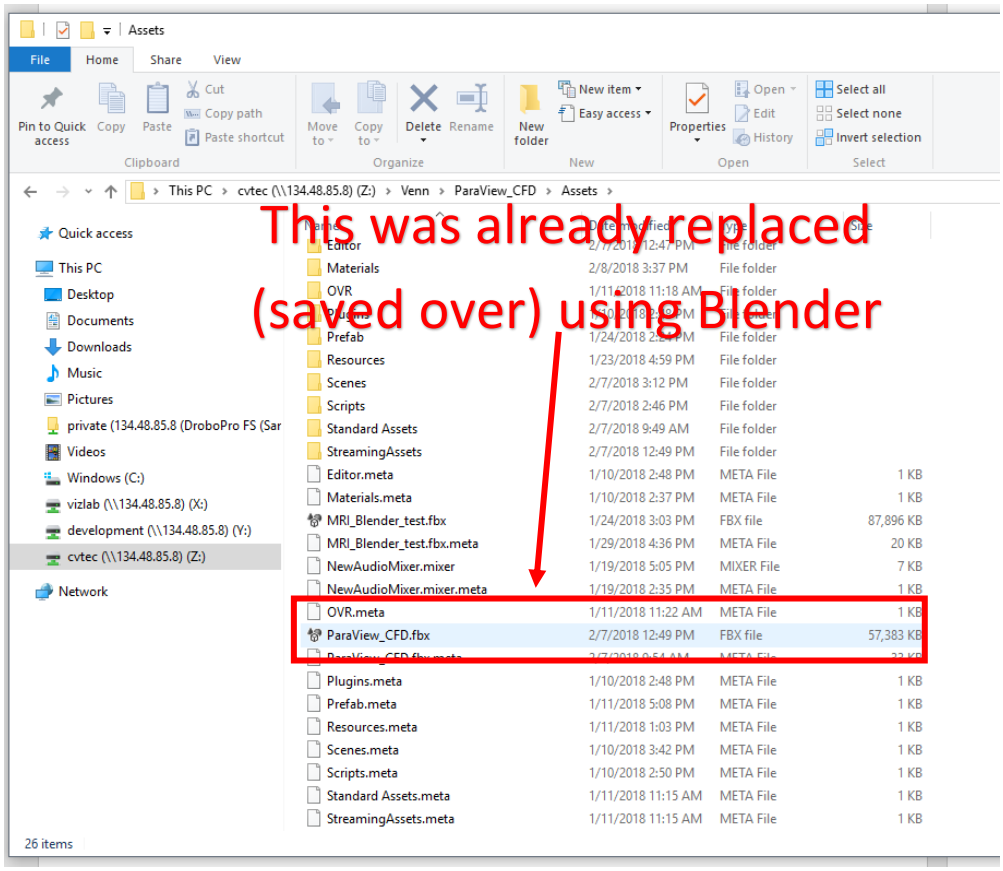

This PC › cvtec (\\134.48.85.8) (Z:) › Venn › ParaView_CFD › Assets
Quick access
Desktop
Downloads
Documents
Pictures
scene
Final CFD to VR scripts-programs-
images used
Phantom- VR test
Workflows
This PC
Desktop
Documents
Downloads
Music
Pictures
private (134.48.85.8 (DroboPro FS (
Name
Date modified
Type
Size
Editor 2/7/2018 12:47 PM File folder
Materials 2/8/2018 3:37 PM File folder
OVR 1/11/2018 11:18 AM File folder
Plugins 1/10/2018 2:48 PM File folder
Prefab 1/24/2018 2:24 PM File folder
Resources 1/23/2018 4:59 PM File folder
Scenes 2/7/2018 3:12 PM File folder
Scripts 2/7/2018 2:46 PM File folder
Standard Assets 2/7/2018 9:49 AM File folder
StreamingAssets 2/7/2018 12:49 PM File folder
Editor.meta 1/10/2018 2:48 PM META File 1 KB
Materials.meta 1/10/2018 2:37 PM META File 1 KB
MRI_Blender_test.fbx 1/24/2018 3:03 PM FBX file 87,896 KB
MRI_Blender_test.fbx.meta 1/29/2018 4:36 PM META File 20 KB
NewAudioMixer.mixer 1/19/2018 5:05 PM MIXER File 7 KB
NewAudioMixer.mixer.meta 1/19/2018 2:35 PM META File 1 KB
ParaView_CFD.fbx 2/7/2018 12:49 PM FBX file 57,383 KB

10) Using File Explorer ( ) Paste each Image you used in the Unity/Assets/Materials/Imaging Data folder – IF THERE ARE ANY IMAGES PREVIOUSLY IN THE FOLDER DELETE THEM FIRST

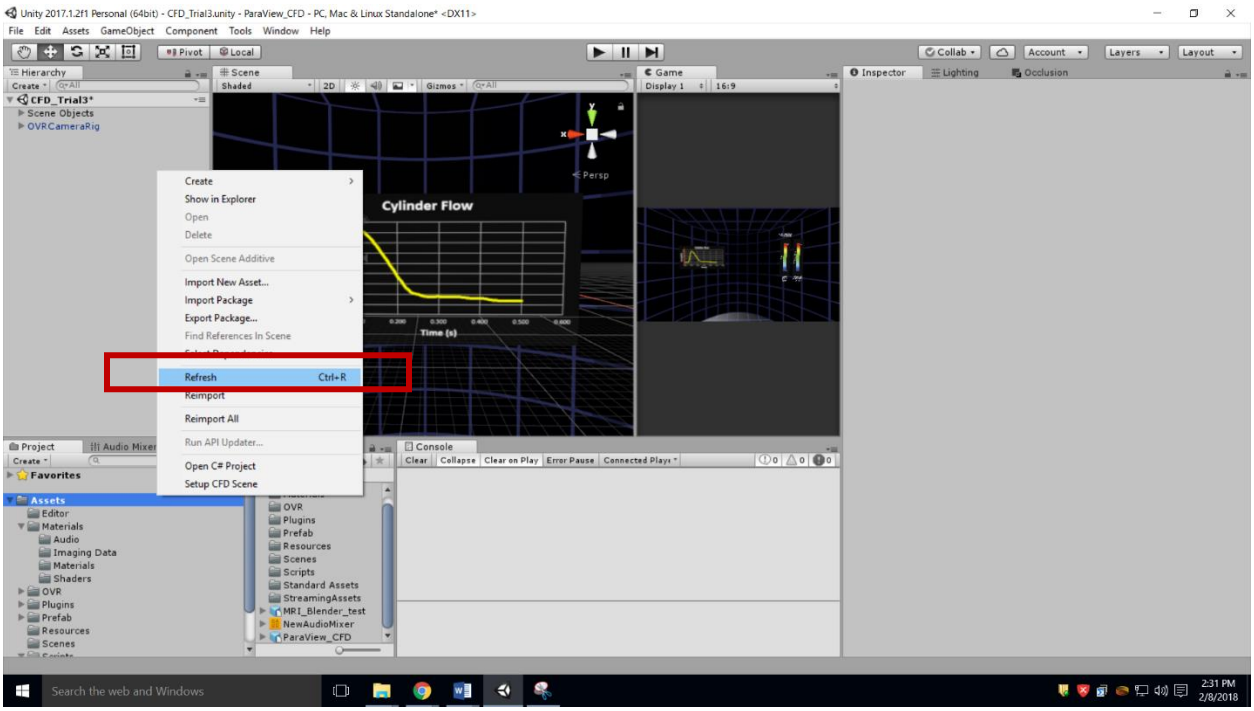


4 Set up CFD model for VR using Unity along with controls

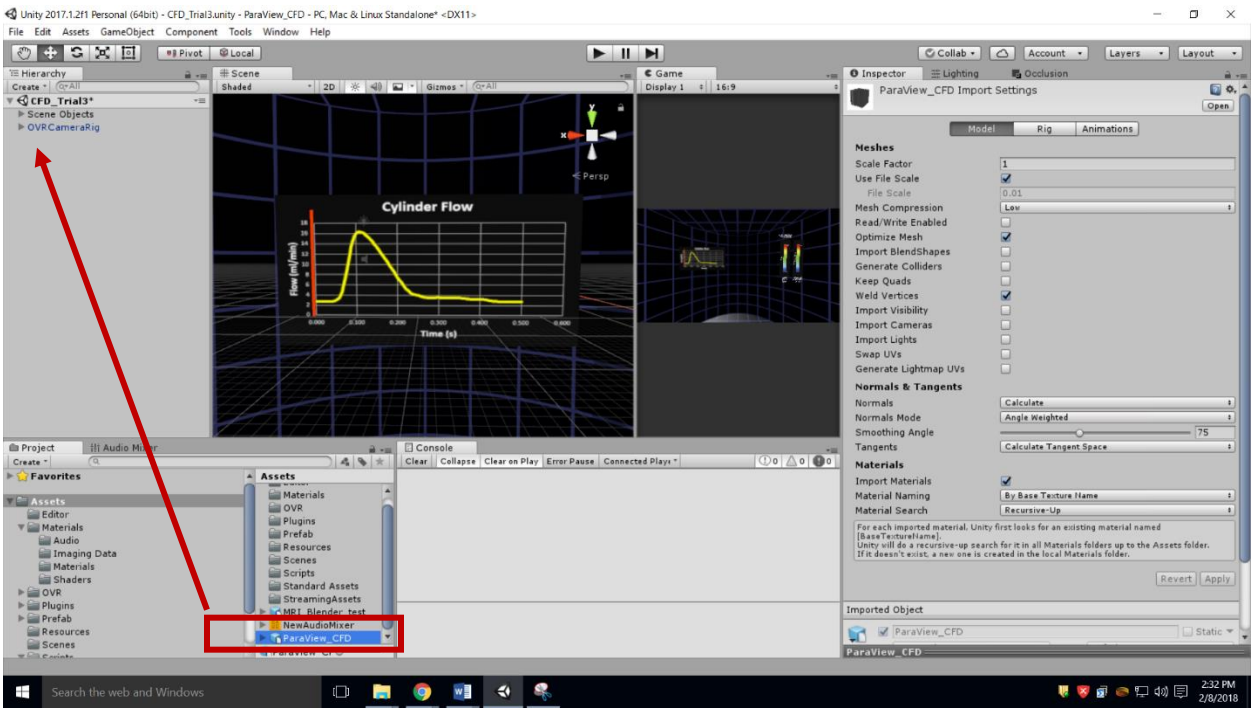
1. This is under the assumption that you already saved over (using Blender) the "ParaView_CFD.fbx" file in the Unity Template Assets folder



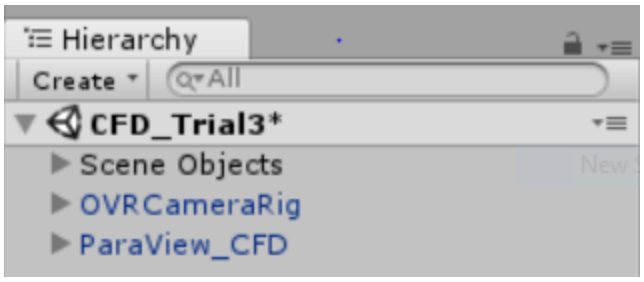
2. Refresh the Unity Scene by right clicking on the Assets folder OR click "Ctrl + R"
    a. This could take up to 20 minutes



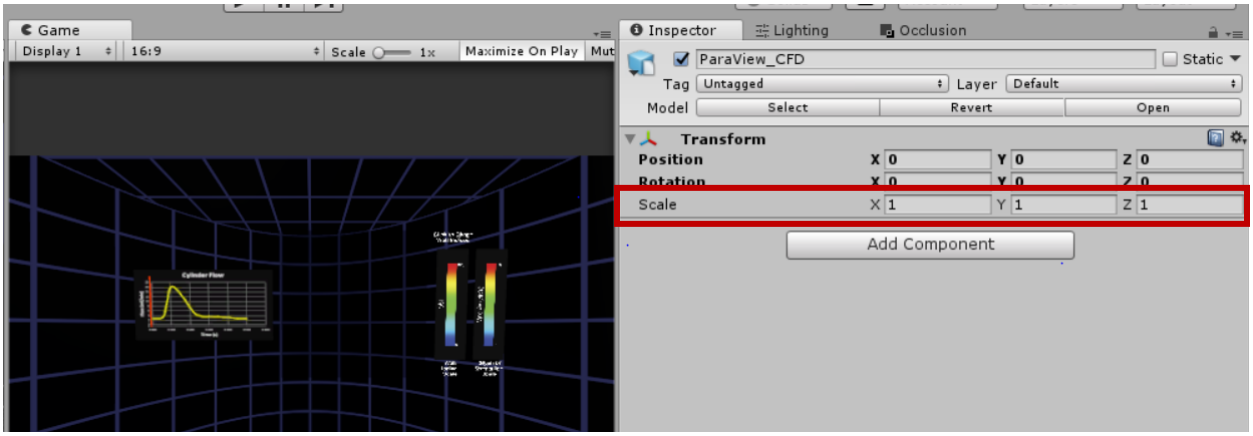
3. Drag and drop the ParaView_CFD.fbx file located in your Assets folder into the Hierarchy



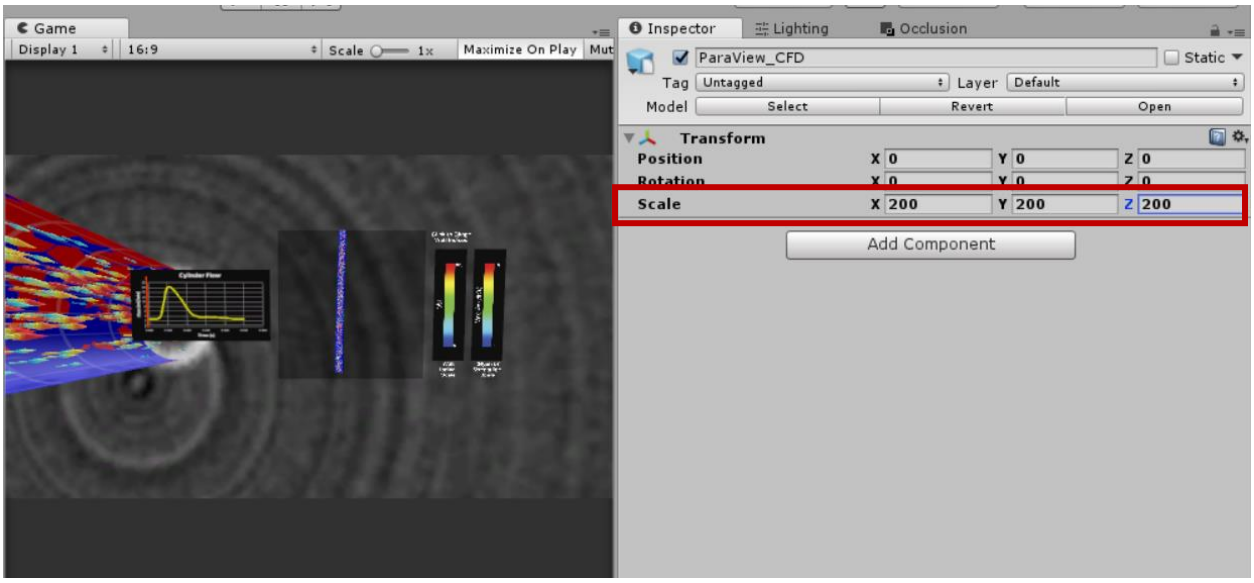
Hierarchy should look like this after dragging and dropping ParaView_CFD into the Hierarchy



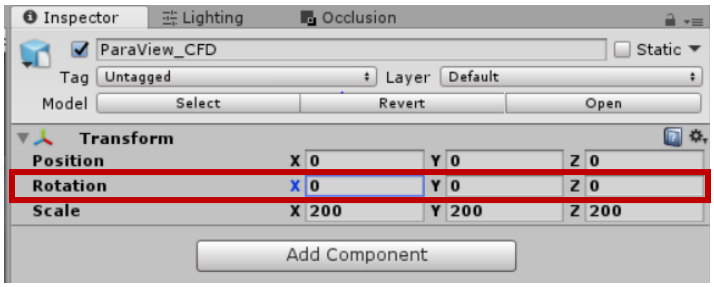
4. Select “ParaView_CFD” in the hierarchy
    a. Scale up the model until it is visible in the Map Camera



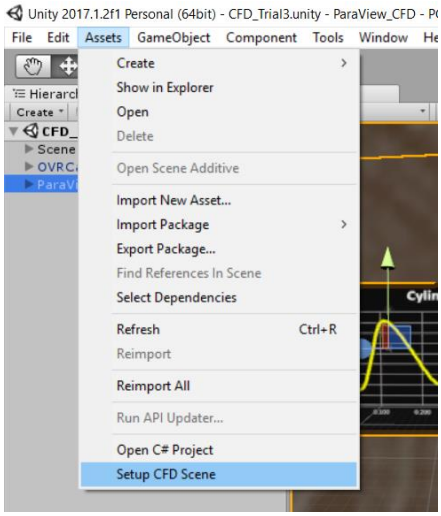
After scaling by 200 notice the model appears in the map view now



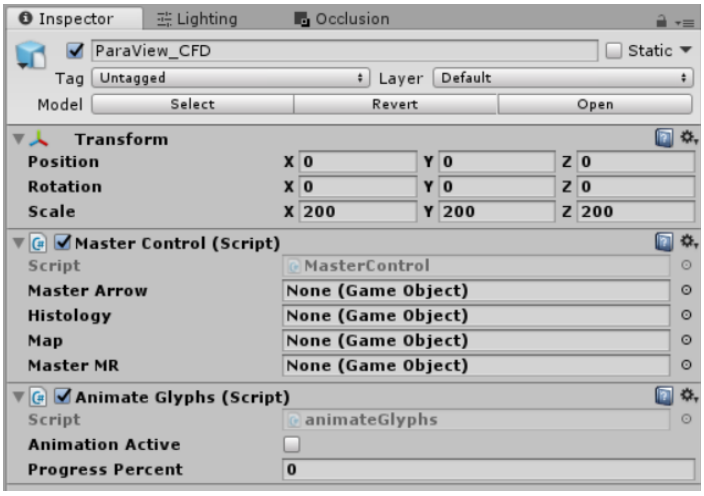
    b. If the model needs to be rotated use the “ParaView_CFD” Rotate X, Y, Z transform to make your model look ideal in the Map View



5. Set up CFD by Selecting Assets → Setup CFD Scene



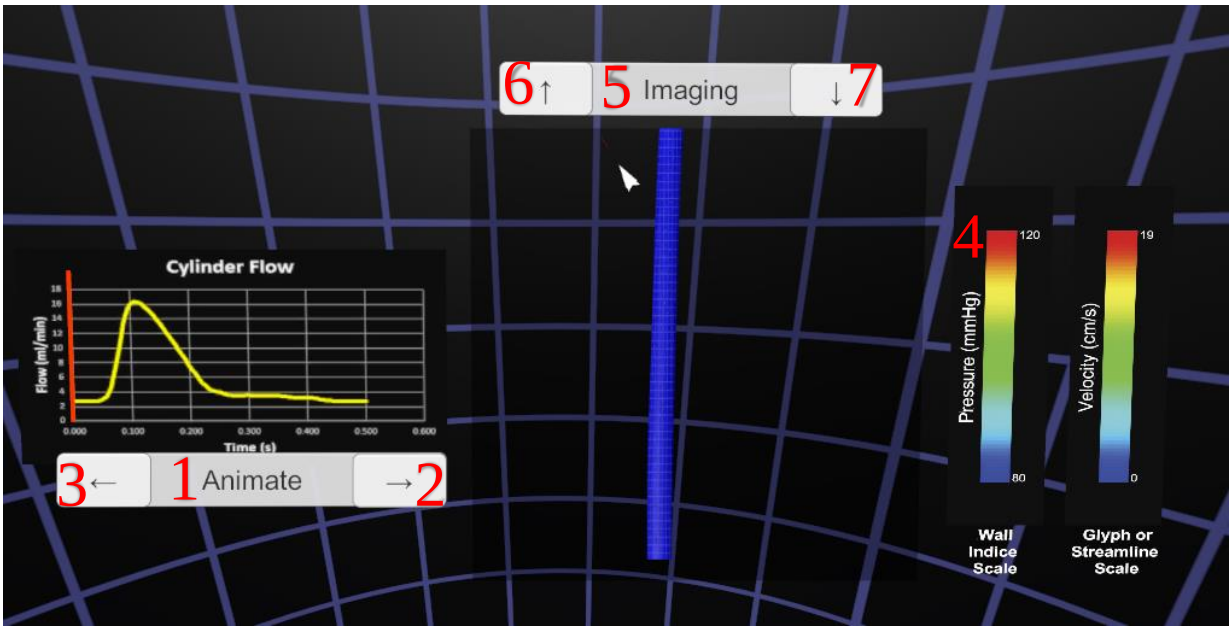
   a. The "ParaView_CFD" Transform should now look like



6. It is now safe to run your model ( ) using the desktop or Oculus Rift
7. Controls
   a. Desktop Version using Keyboard
      i. Play/Pause animation: Space Bar

ii. While in Pause mode, move forward one time step: "→"
iii. While in Pause mode, move back one time step: "←"
iv. Switch Wall Indices: "Z"
v. Turn on/off Volumetric Imaging: "M"
vi. While in Image mode, move up one Image Slice: "↑"
vii. While in Image mode, move down one Image Slice: ↓
viii. Movement Forward: "W"
ix. Movement Backward: "S"
x. Movement Left: "A"
xi. Movement Right: "D"
xii. Steering: Use Mouse

b. Desktop Version using Clicking
   i. Use Left Mouse Click on select Buttons/Scales
      1. Play/Pause animation
      2. While in Pause mode, move forward one time step
      3. While in Pause mode, move back one time step
      4. Switch Wall Indices
      5. Turn on/off Volumetric Imaging
      6. While in Image mode, move up one Image Slice
      7. While in Image mode, move down one Image Slice
   ii. Movement Forward: "W"
   iii. Movement Backward: "S"
   iv. Movement Left: "A"
   v. Movement Right: "D"
   vi. Steering: Use Mouse



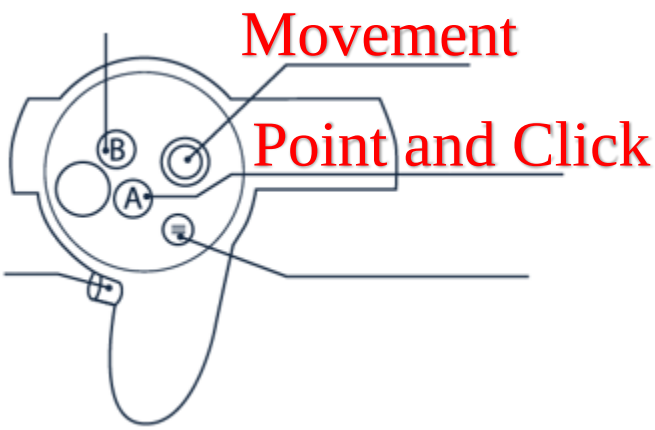
c. X-Box Controls using Oculus Rift
   i. Click on X-Box controller buttons/analog stick

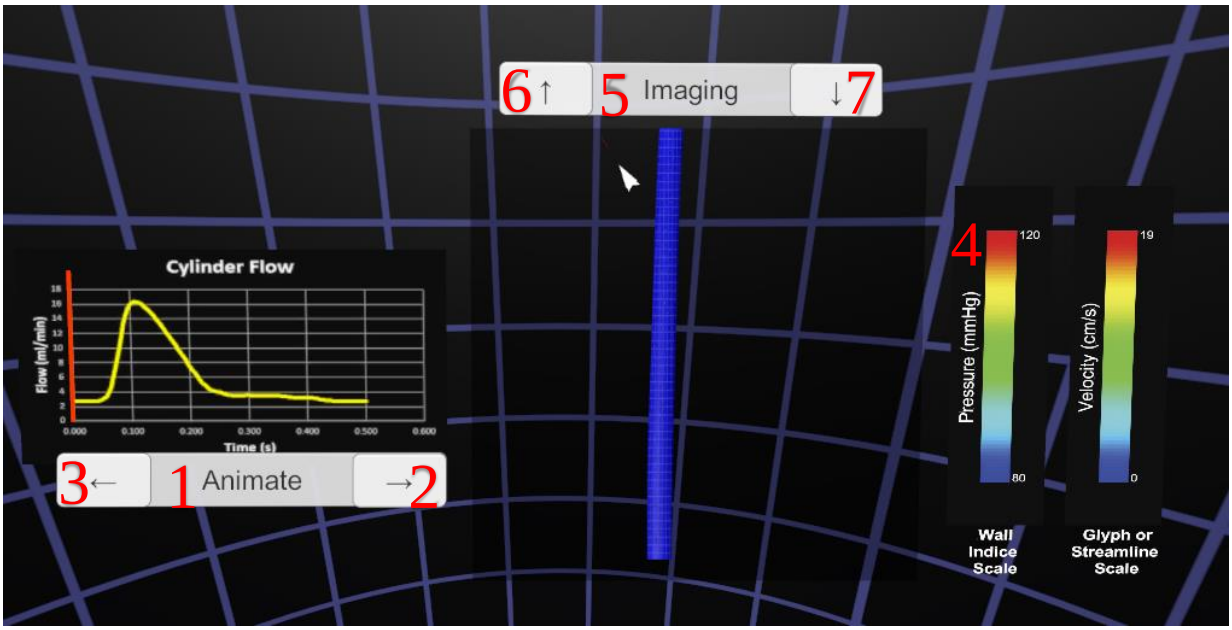
1. Play/Pause animation
2. While in Pause mode, move forward one time step
3. While in Pause mode, move back one time step
4. Switch Wall Indices
5. Turn on/off Volumetric Imaging
6. While in Image mode, move up one Image Slice
7. While in Image mode, move down one Image Slice
8. Movement (Backward/Forward/Left/Right)

ii. Steering: Your Head rotation

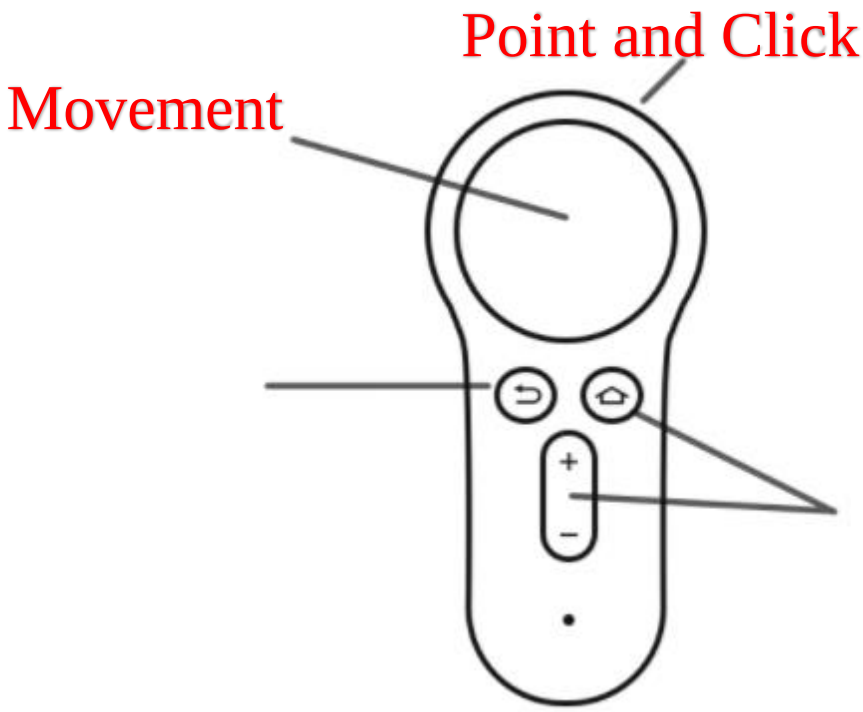
d. Using Oculus Rift Trackable controllers
   i. Use Right Controller to point wand to the intractable buttons/scales then press the “A” button for execution
      1. Play/Pause animation
      2. While in Pause mode, move forward one time step
      3. While in Pause mode, move back one time step
      4. Switch Wall Indices
      5. Turn on/off Volumetric Imaging
      6. While in Image mode, move up one Image Slice
      7. While in Image mode, move down one Image Slice
   ii. Movement: Right controller Analog stick

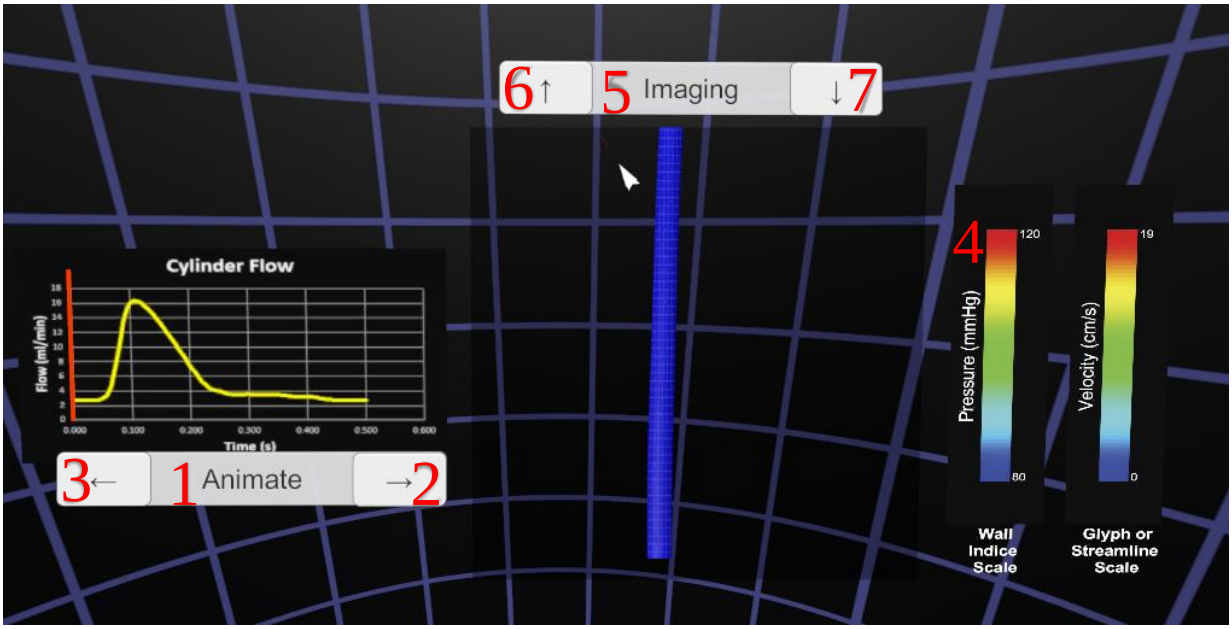
iii. Steering: Your Head rotation





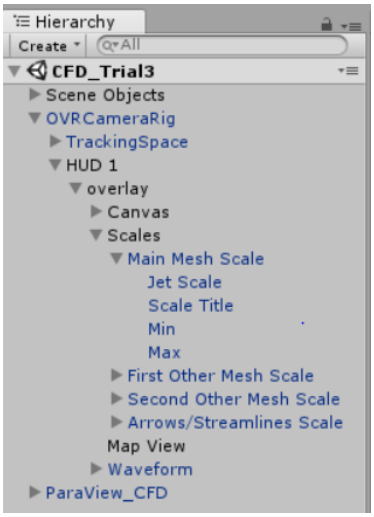
e. Using Gear VR Controller
   i. Use Gear VR Controller to point wand to the intractable buttons/scales then press the back trigger button for execution
      1. Play/Pause animation
      2. While in Pause mode, move forward one time step
      3. While in Pause mode, move back one time step
      4. Switch Wall Indices
      5. Turn on/off Volumetric Imaging
      6. While in Image mode, move up one Image Slice
      7. While in Image mode, move down one Image Slice
   ii. Movement: Touchpad
   iii. Steering: Your Head rotation


Point and Click
Movement

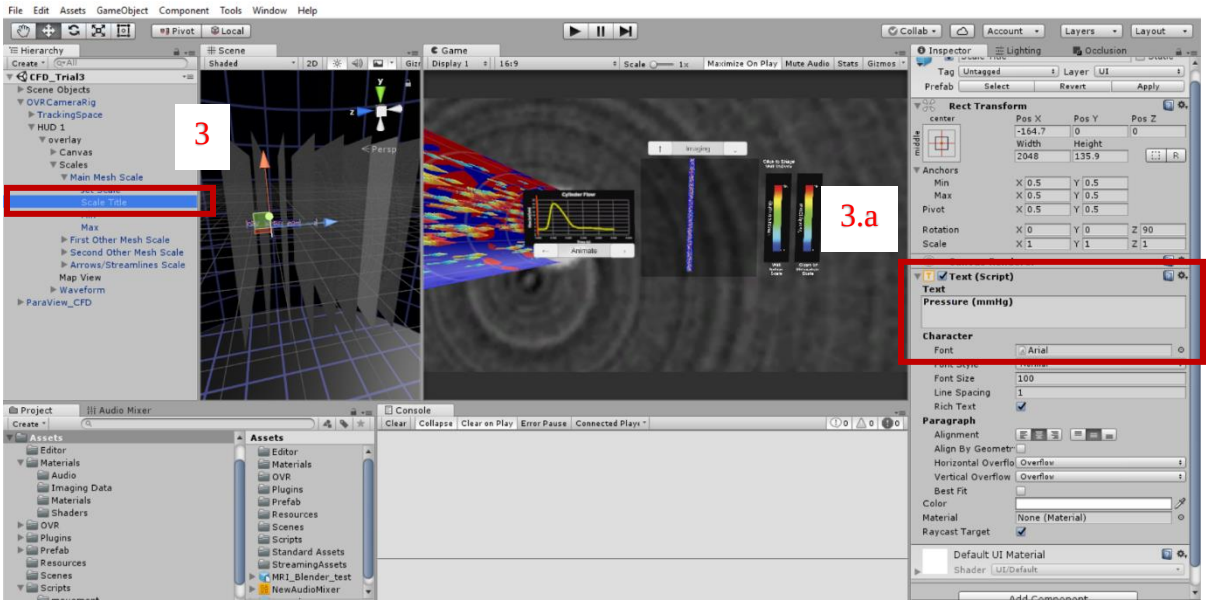

6↑
5 Imaging
↓7
Cylinder Flow
Flow (ml/min)
Time (s)
3←
1 Animate
→2
4
120
80
Pressure (mmHg)
Wall Indice Scale
19
0
Velocity (cm/s)
Glyph or Streamline Scale

4.1 Label scales using Unity

1. These steps are done for each of your Scales (Main Mesh Scale, First Other Mesh Scale, Second Other Mesh Scale, and/or Arrows/Streamlines Scale)
    a. If you do not have “Other Mesh” disregard the “First Other Mesh Scale” and “Second Other Mesh Scale”



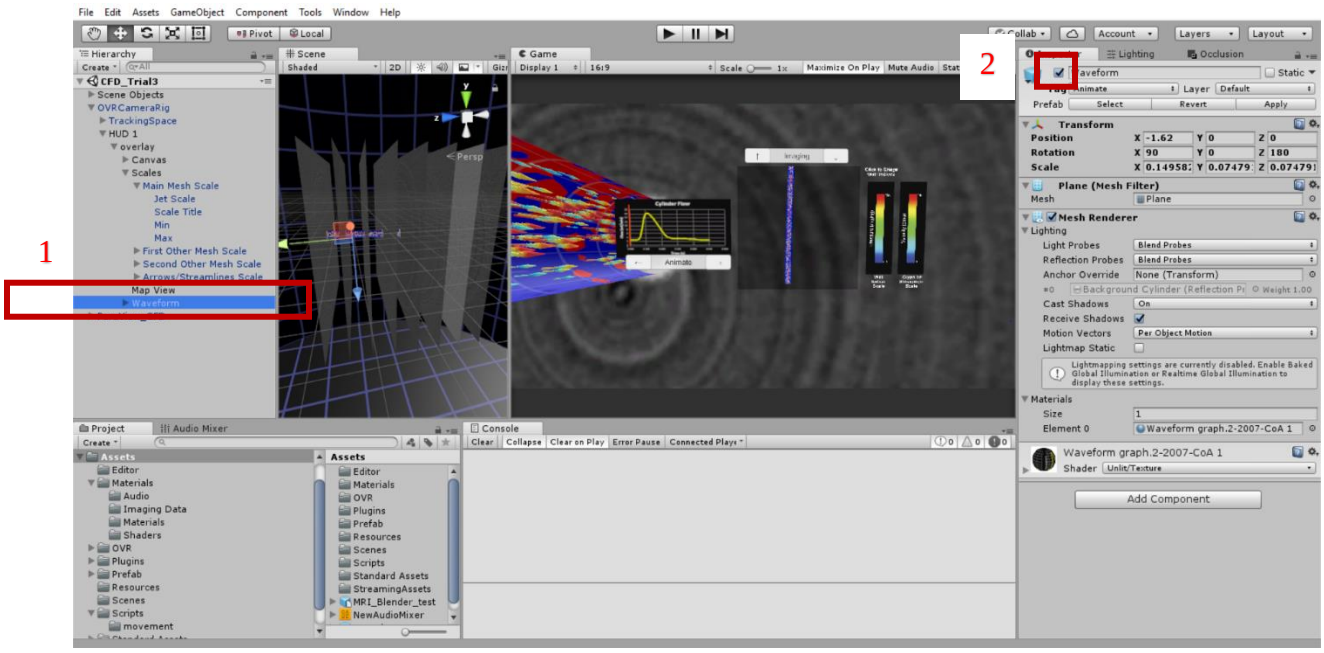
2. Click on your Scale under OVRCameraRig/HUD 1/Scales/WHATEVER SCALE
3. Click on “Scale Title”
    a. In the Transform, in Text (Script) type in the Title of your scale with appropriate units
4. Click on “Min”
    a. In the Transform, in Text (Script) type in the minimum number of your scale
5. Click on “Max”
    a. In the Transform, in Text (Script) type in the maximum number of your scale



4.2 Add/remove flow waveform using Unity

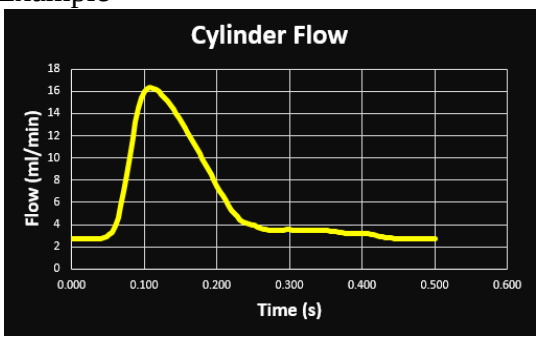
4.2.1 Turn off flow waveform

1. Select the Waveform game object under OVRCameraRig/HUD 1/Overlay/Waveform in the Hierarchy
2. In the Waveform Inspector, uncheck the box next to the right of the blue cube and above the word “Tag”

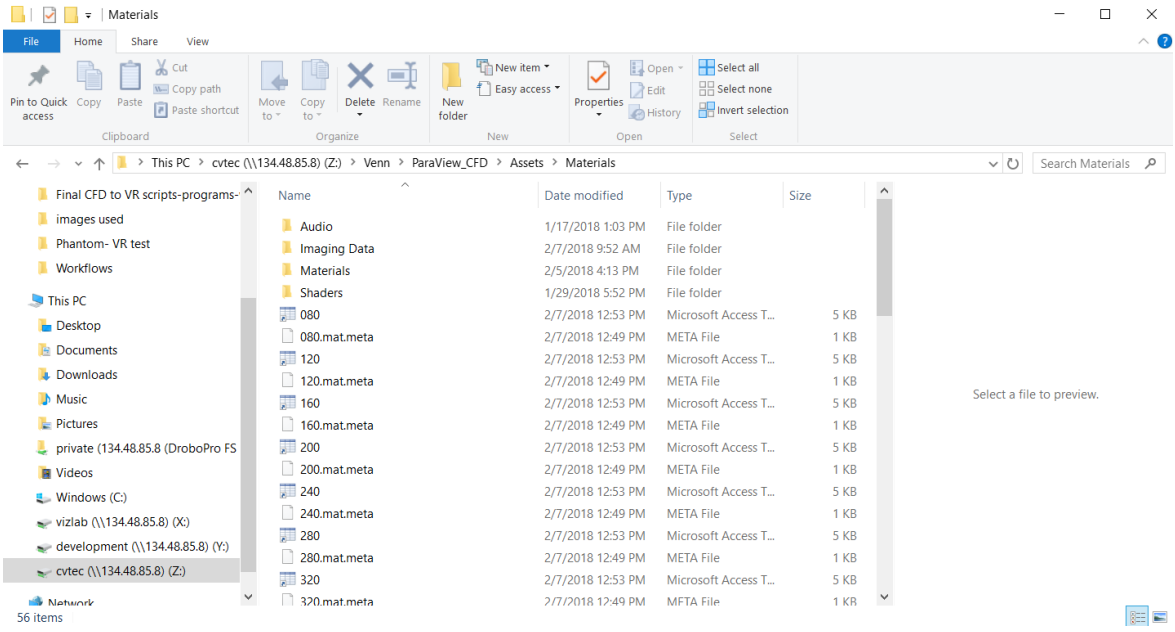


4.2.2 Create Animated Flow waveform

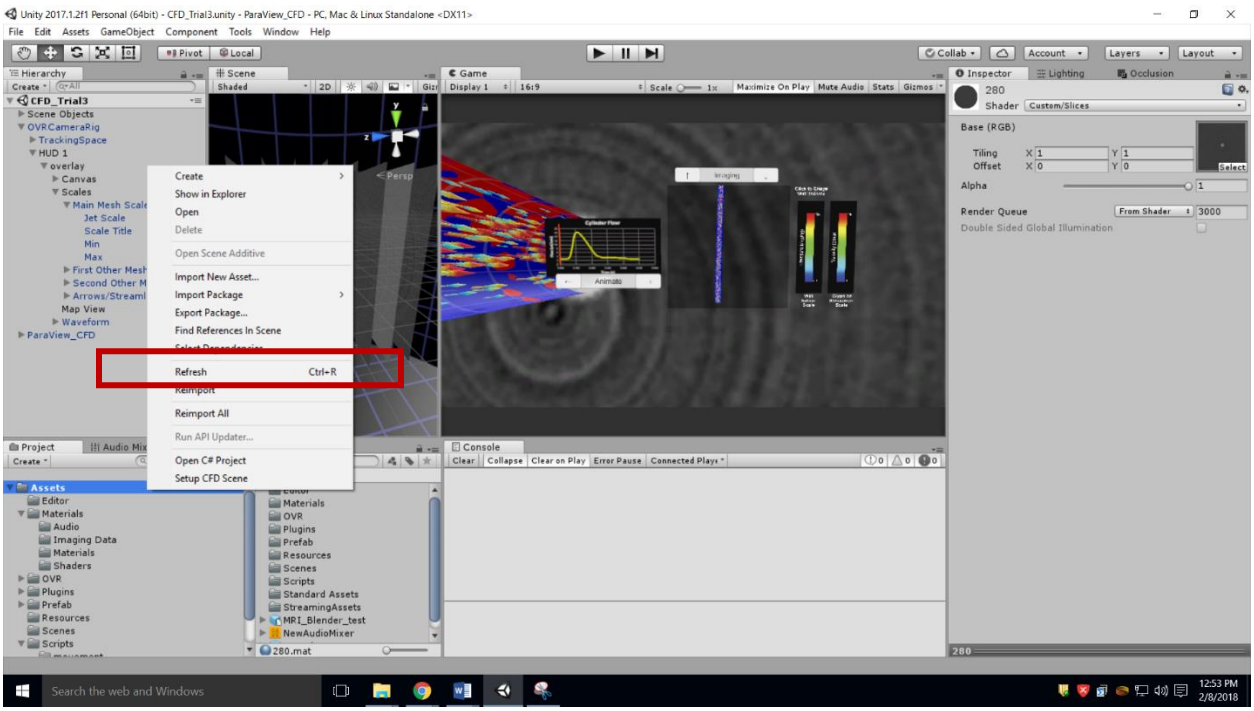
1. Create a flow waveform for your CFD simulation
2. Save the waveform as a PNG image
    a. Example



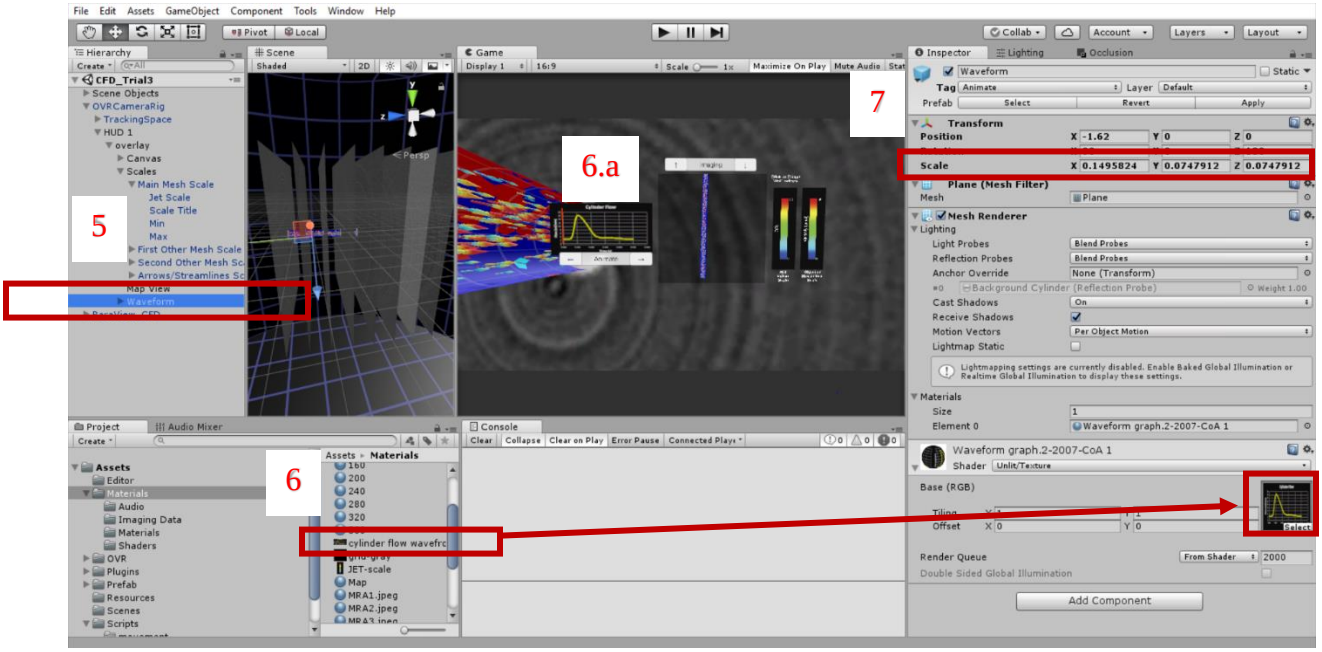
3. Using File Explorer () Paste the PNG image in the Unity Template Materials folder



4. Refresh the Unity Scene by right clicking on the Assets folder OR click “Ctrl + R”



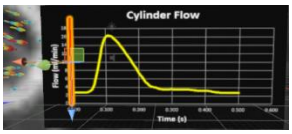
5. Select on the Waveform game object in the Hierarchy
6. In the M aterials folder, drag and drop the waveform PNG image into the shader texture box
    a. Your flow waveform should now appear in the Unity scene
7. The scale of the waveform might need to be adjusted- to do this adjust Transform Scale X and Z



8. Select the Indicator object underneath the Waveform in the Hierarchy

a. The line on your waveform should appear orange in the Unity scene

9. Adjust the Transform, Position X of the Indicator to align with the start of the flow waveform
    a. Copy the Position X
    b. Past the Position X value to the Move Indicator Script Min X slot



10. Adjust the Transform, Position X of the Indicator to align with the end of the flow waveform
    a. Copy the Position X
    b. Past the Position X value to the Move Indicator Script Max X slot