Kuipers Bed Post Processing

February 2018

CPFD Software LLC
10899 Montgomery Blvd. NE, Suite A
Albuquerque, NM 87111
+1.505.275.3849
www.cpfd-software.com

Outline

- The post-processing section is split into three parts:
  - Basic post-processing
  - Advanced post-processing
  - Post-processing assignments

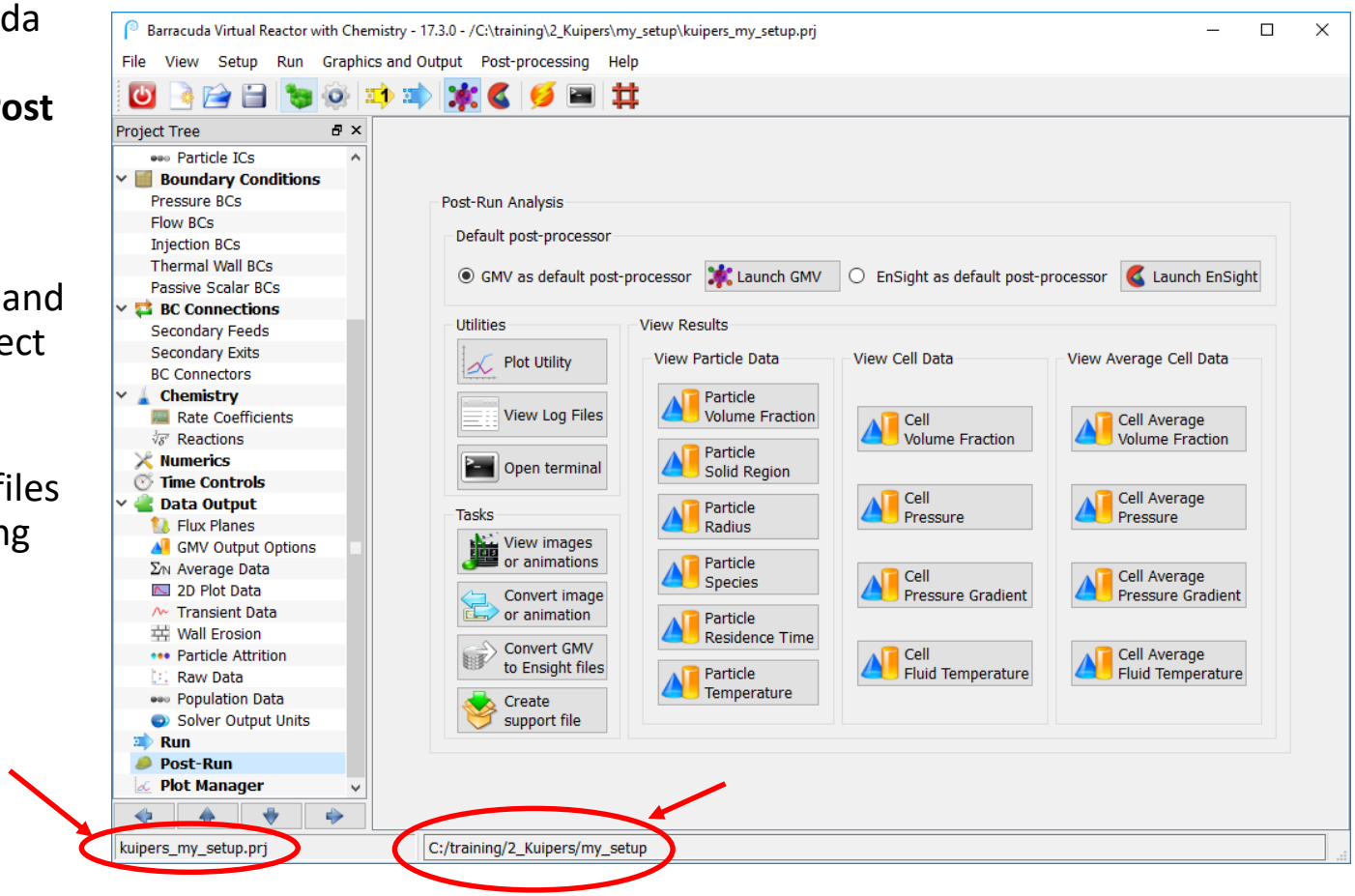
Part I: Basic Post-Processing

Training Objectives:

- Overview of GMV shortcut buttons
- GMV Basics
  - Changing GMV view angle
  - Viewing a subset of the data
  - Setting data limits
  - Adjusting background and material colors
  - Adding titles
- Saving attribute files
- Creating and viewing images
- Creating and viewing animations
- Plotting transient data in XMGR

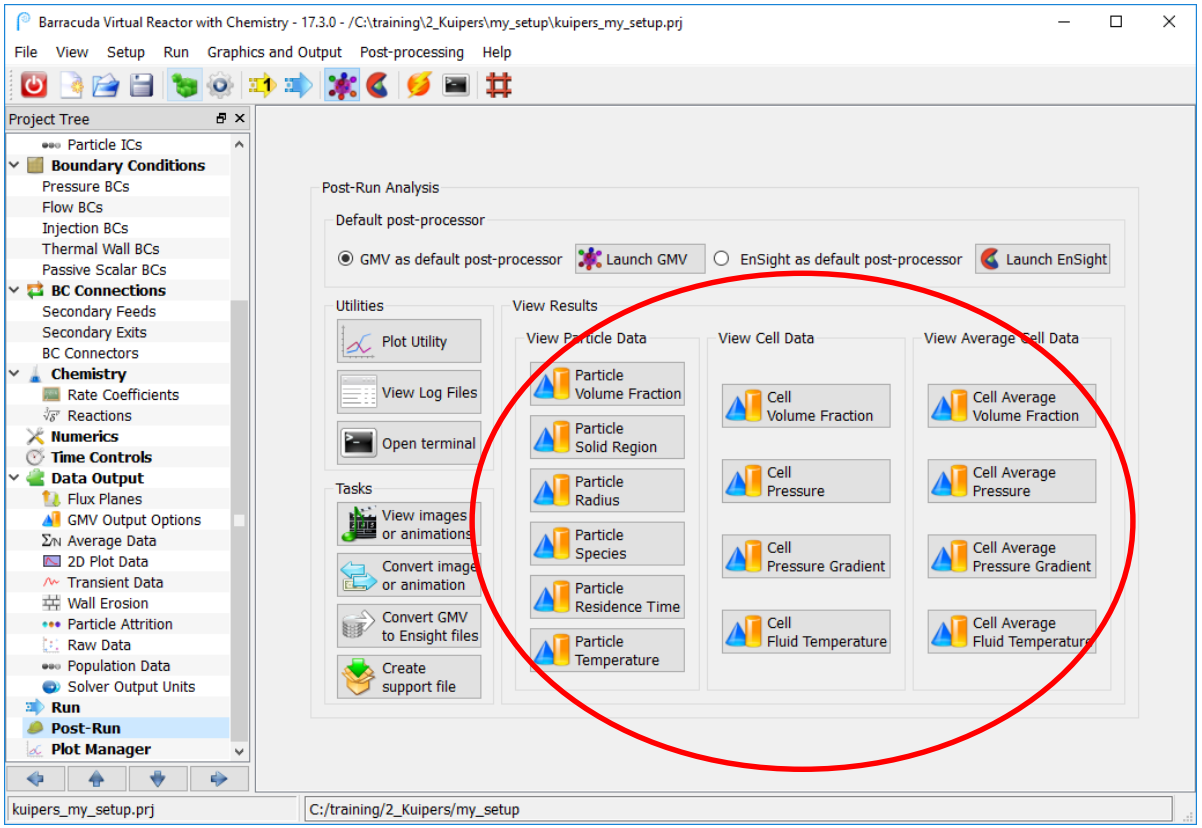
Post-Run

- During and after a Barracuda simulation, examine your results by clicking on the **Post Run** tab
- Check that the project file and working directory are correct
- **Note:** You can view GMV files while the solver is operating



GMV Shortcut Buttons

- There are several ways to access and analyze the simulation results
- The shortcut buttons shown on the right provide convenient access to common views of results
- **Tip:** Not all shortcut buttons are available for all simulations. If the data is not available, Barracuda will warn you (e.g. clicking on average data buttons, when averaging has not been set)

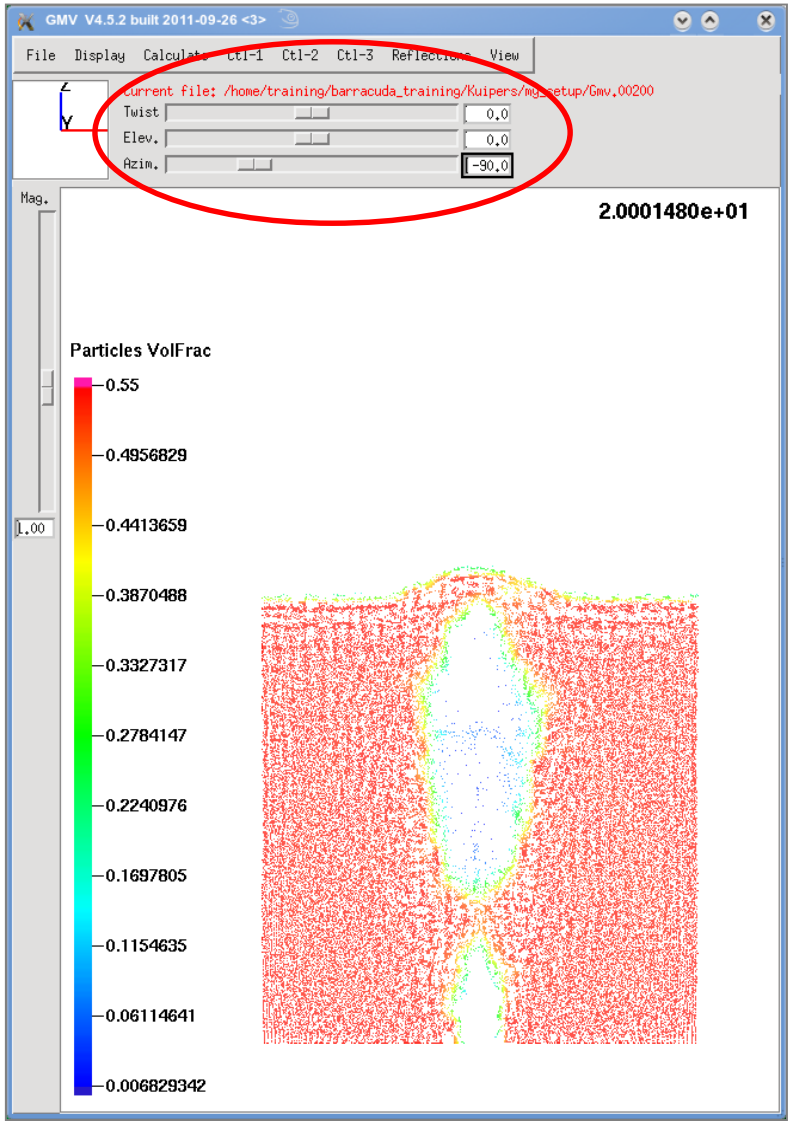


Adjusting Model View

- Click on **Particle Volume Fraction**

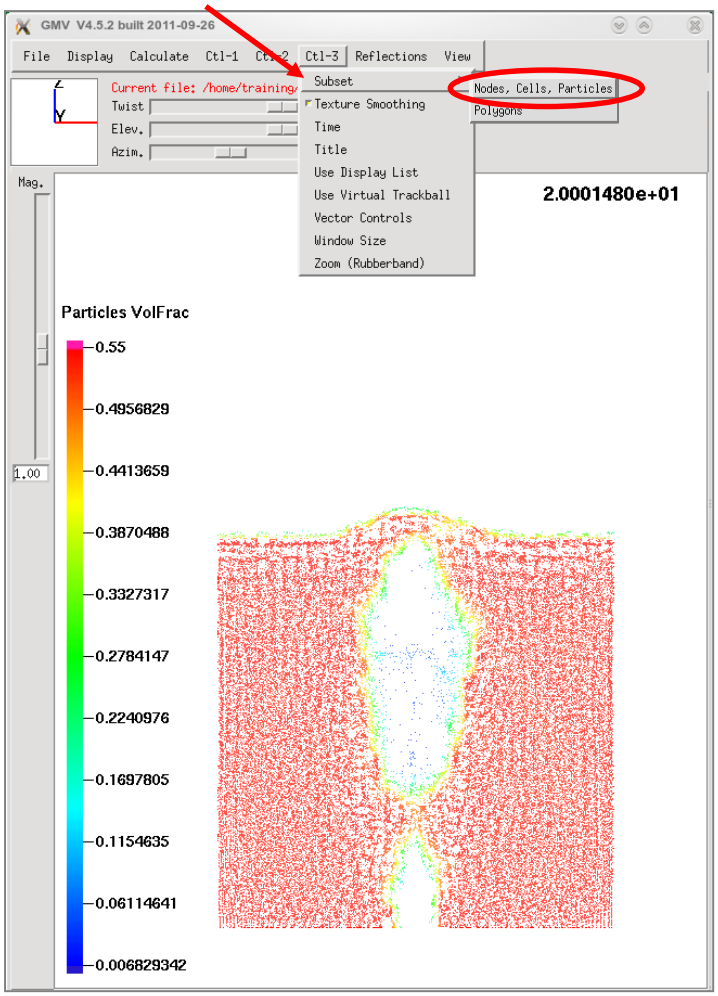
- A GMV window will pop up showing particles colored by volume fraction

- The default view angle makes the model hard to see, so turn the model face-on (angle 0, 0, -90)
  - You may use your mouse, the slider bars, or enter the values in directly



Viewing a Subset

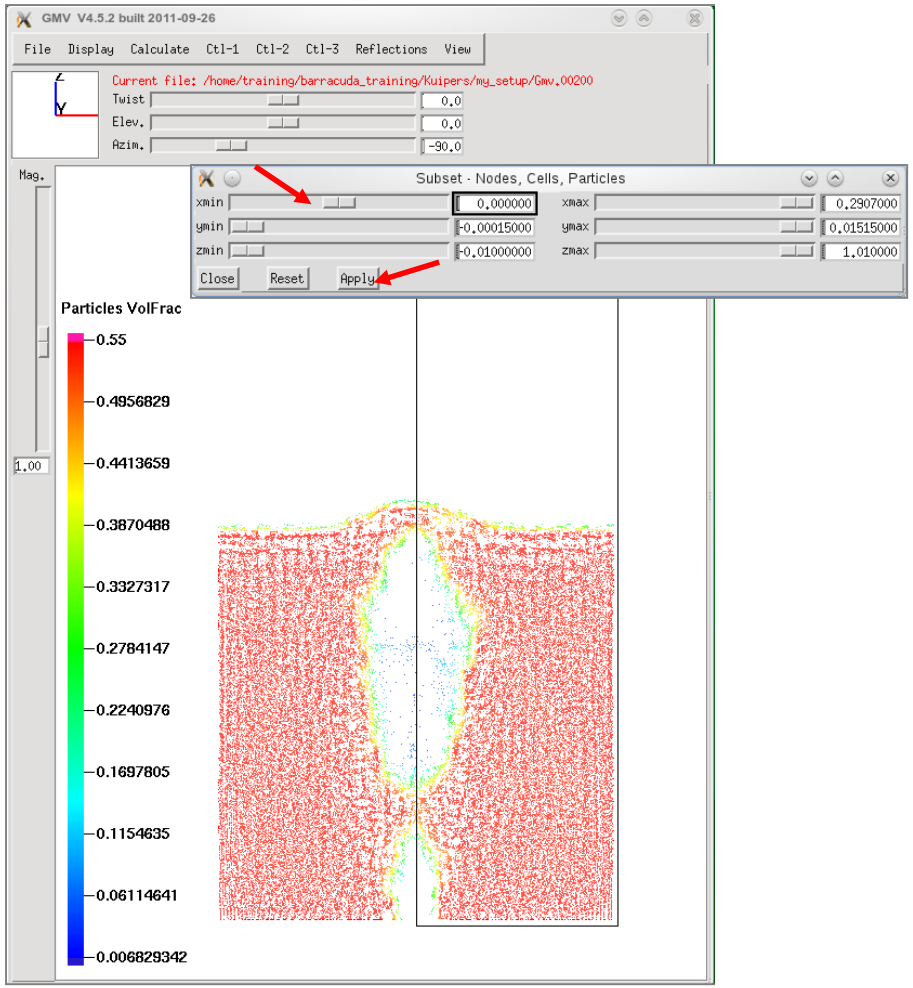
- A subset allows the viewing of smaller sections of the full model
- This is commonly used for looking inside a large model
- Select **Ctl-3 → Subset → Nodes, Cells, Particles** to raise the Nodes, Cells, Particles Subset window



Viewing a Subset

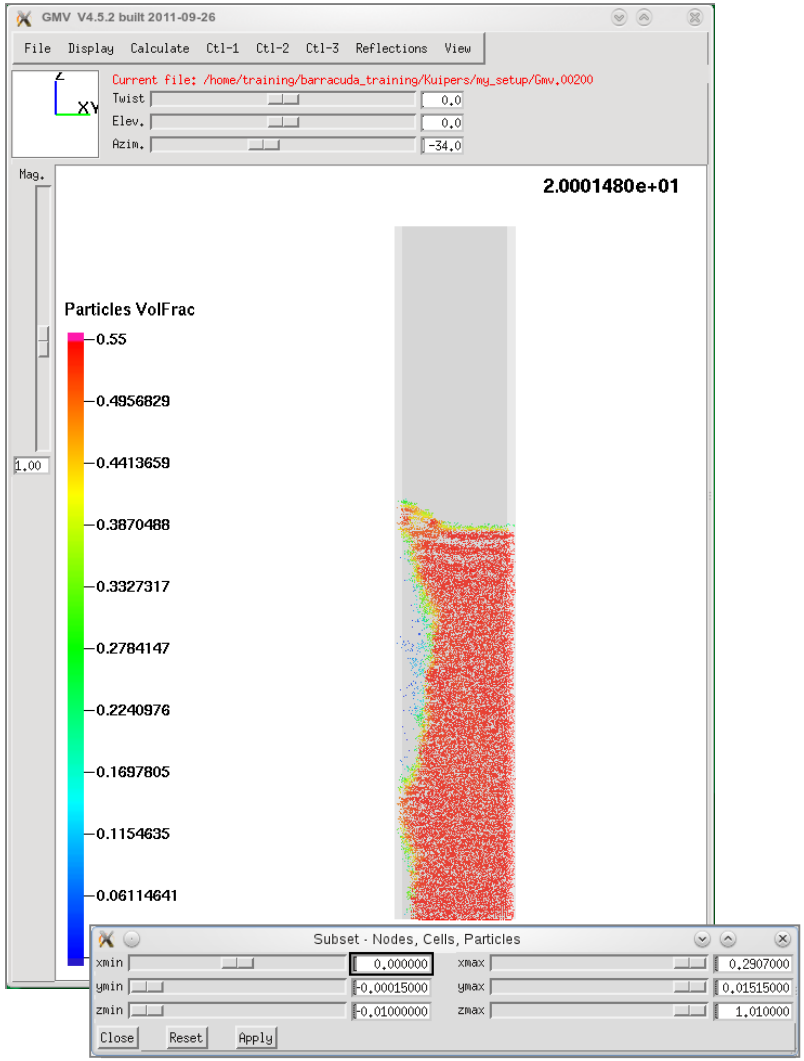
- Move the **x-min** slider so that half of the model is selected

- Click **Apply** in the Nodes, Cells, Particles Subset window



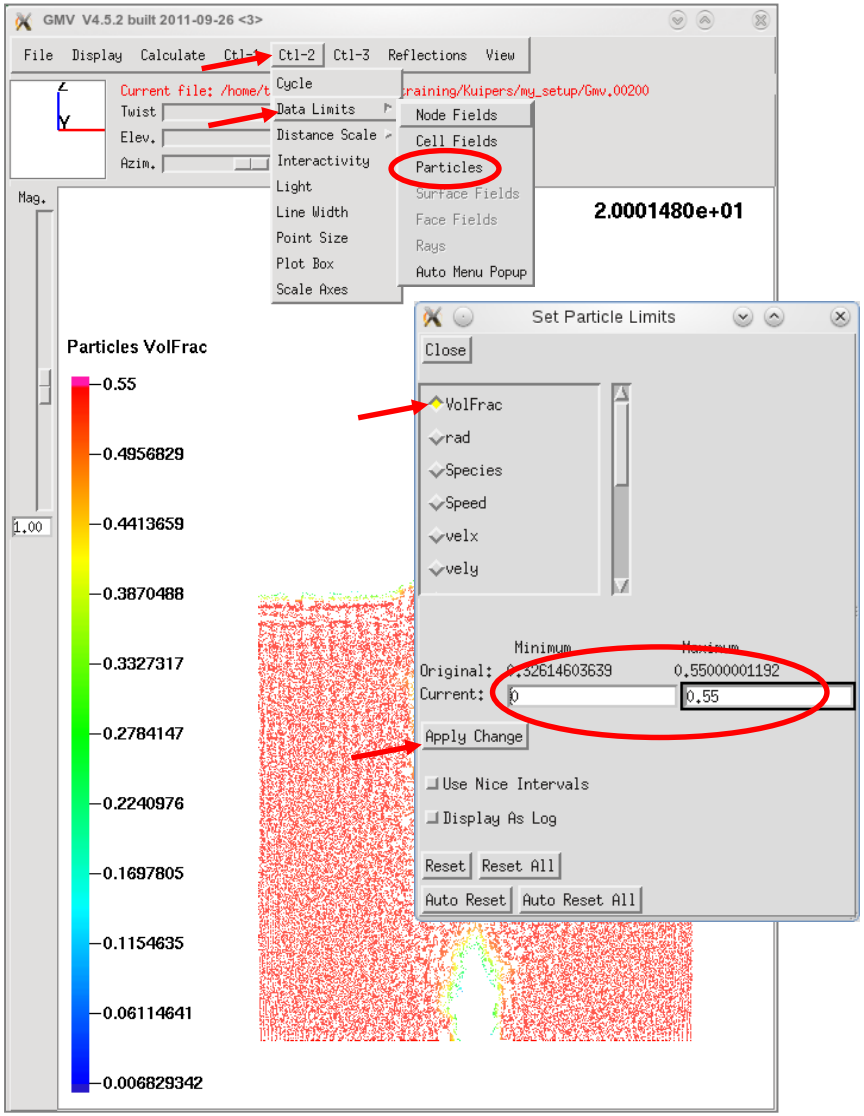
Viewing a Subset

- Rotate the model to view the inside
- Is anything different happening on the inside that cannot be seen from the whole model view?
- Not in this case, move the **x-min** back to the left and click **Apply**
- **Hint:** Always view results from both the inside and outside of any model to view any hidden behavior
- Reset the view angle to (0,0,-90)



Setting Data Limits

- You can set the data limits of the particles volume fraction
- Select **Ctl-2 → Data Limits → Particles**
- A new window will pop up. Set the VolFrac data limits from
  "0" to "0.55"
- Click on **Apply Change**
  - Notice the change in the color scale values. Values are now in nice, clean intervals



Opening a Gmv

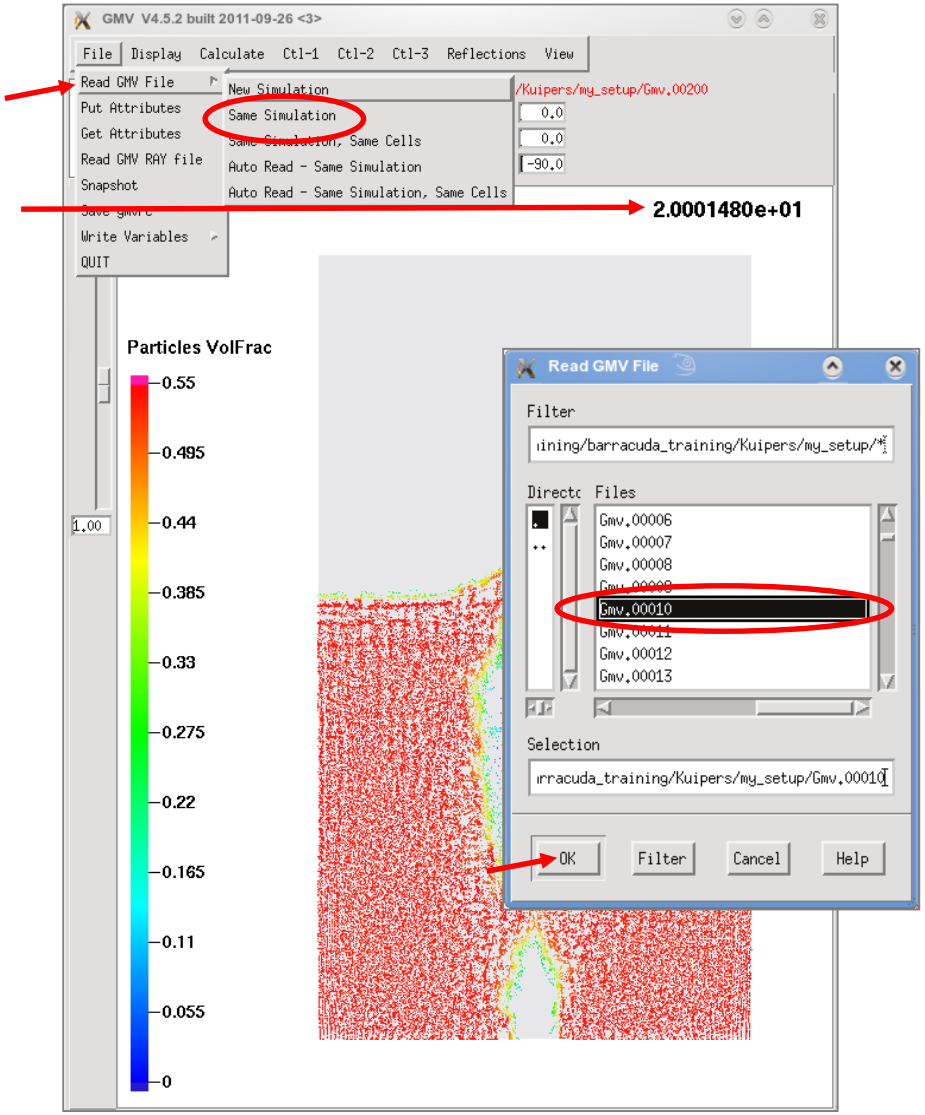
- Note that when a button on the Barracuda post-run page is clicked, GMV will open the last Gmv* file written for the simulation. In this case, this corresponds to the simulation end time of 20 seconds (if the calculation is complete)

- To view a file for a different point in time, Select **File → Read GMV File → Same Simulation**

- Select the Gmv file that corresponds to 1 sec into the simulation. Remember in the problem setup, the plot interval was set to 0.1 sec, so Gmv.00010 corresponds to 1 sec.

- Click **OK**

- Next, open the Gmv file that corresponds to 15 sec into the simulation



Additional Gmv Shortcuts

- Use the **View Results** buttons to examine your simulation results

- Do the results reflect the behavior you expected?

  _____

  _____

- How does the average volume fraction compare to the instantaneous volume fraction?

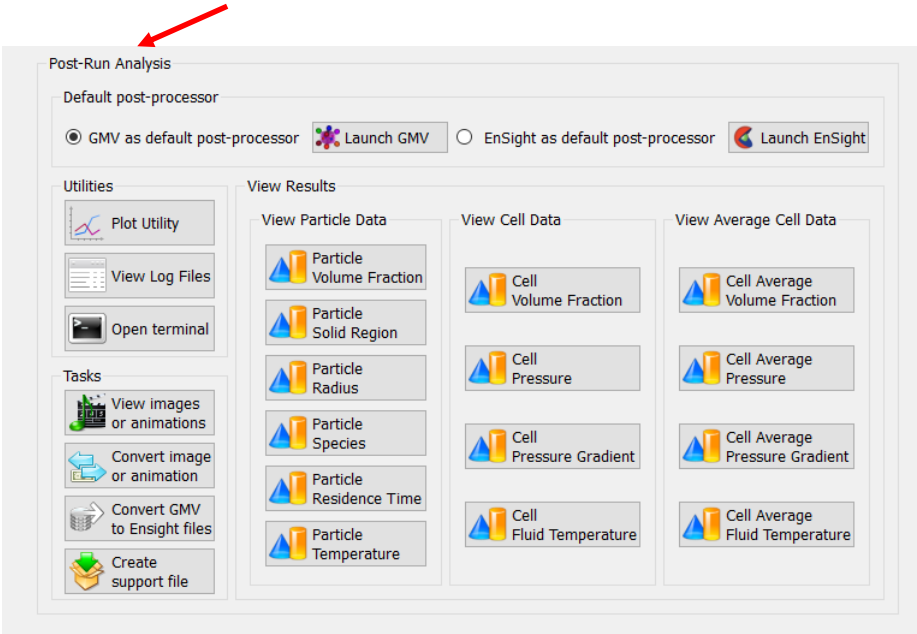
  _____

  _____

- How does the cell volume fraction compare to the particle volume fraction?

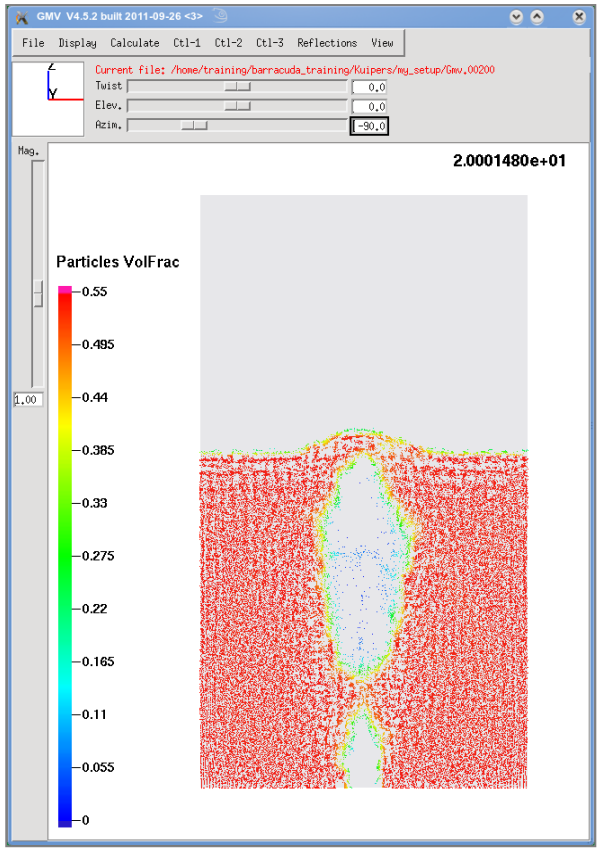
  _____

  _____

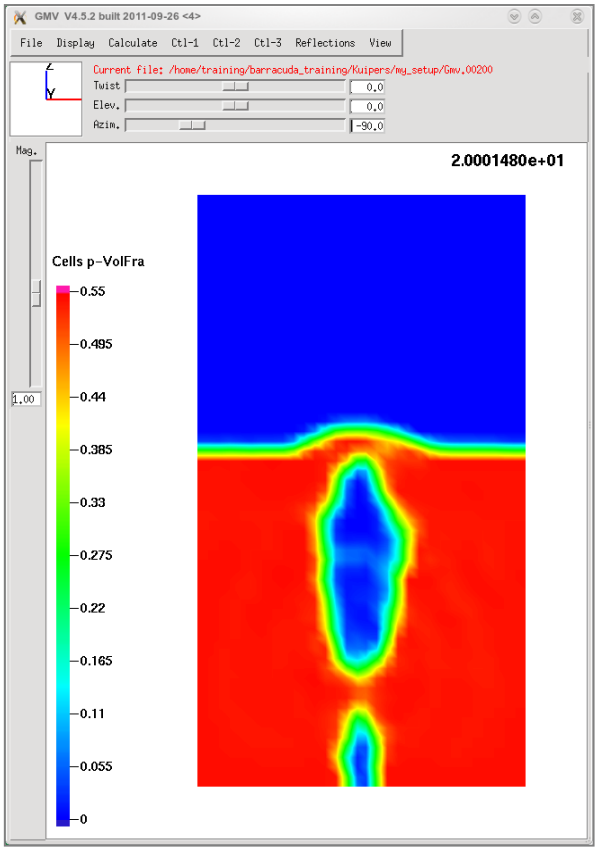


Comparing Results

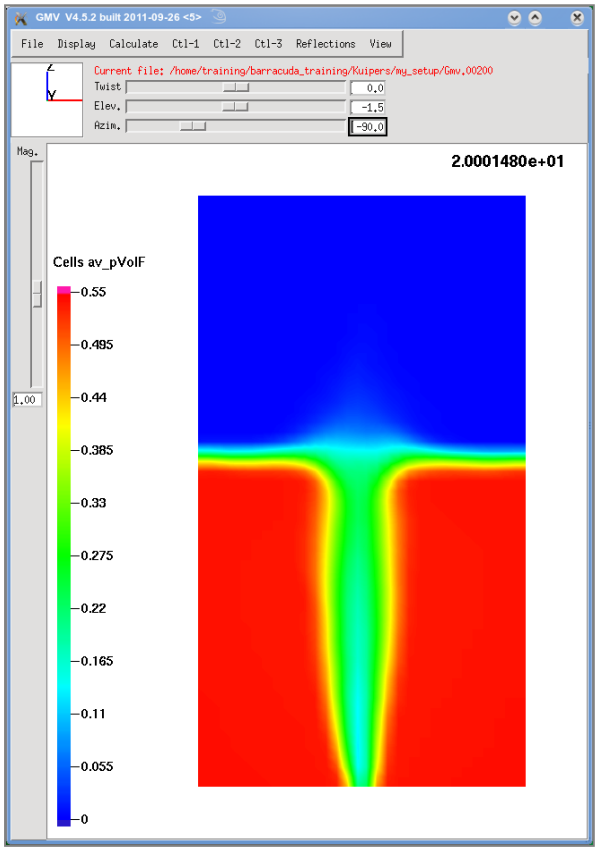
Particle Volume Fraction



Cell Volume Fraction

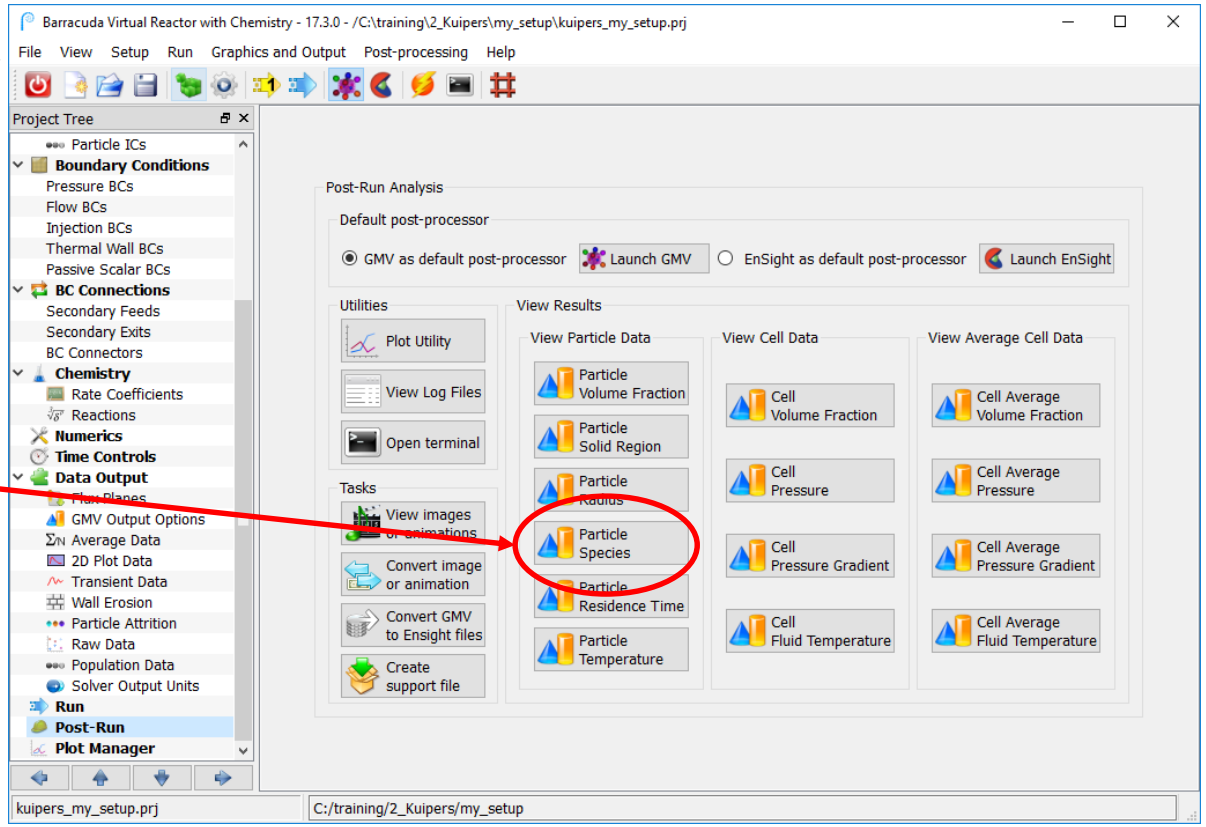


Average Volume Fraction



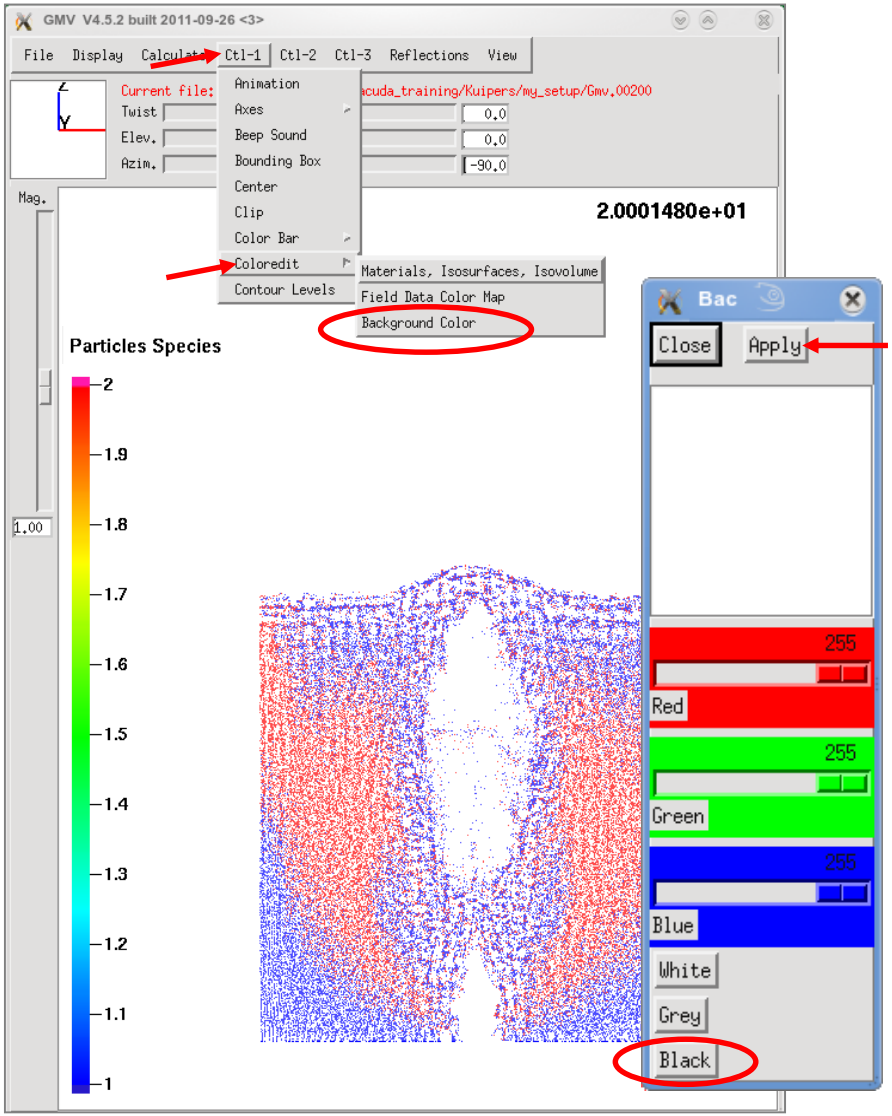
Particle Mixing in Kuipers Bed

- GMV and Barracuda can be used to make high quality images and animations. Colors, labels, and other settings can be adjusted to best display the simulation results
- To demonstrate, an animation showing the particle mixing will be created
- Click on **Particle Species**



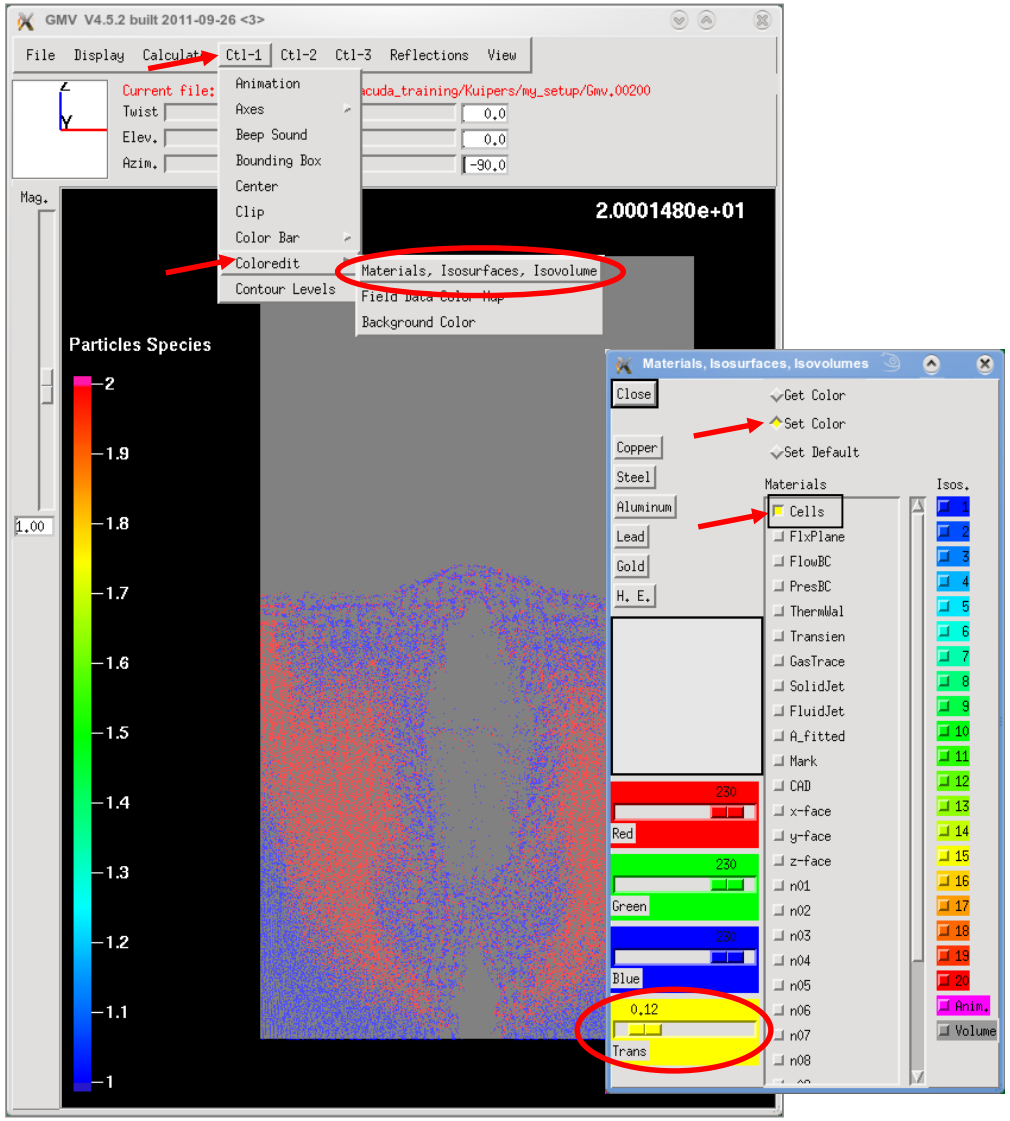
Background Color

- Rotate the image into view (angle 0, 0, -90)
- Animations often look nice with a black background
- Select **Ctl-1 → Coloredit → Background Color** to raise the Background Color window
- Select **Black**
- Click **Apply**



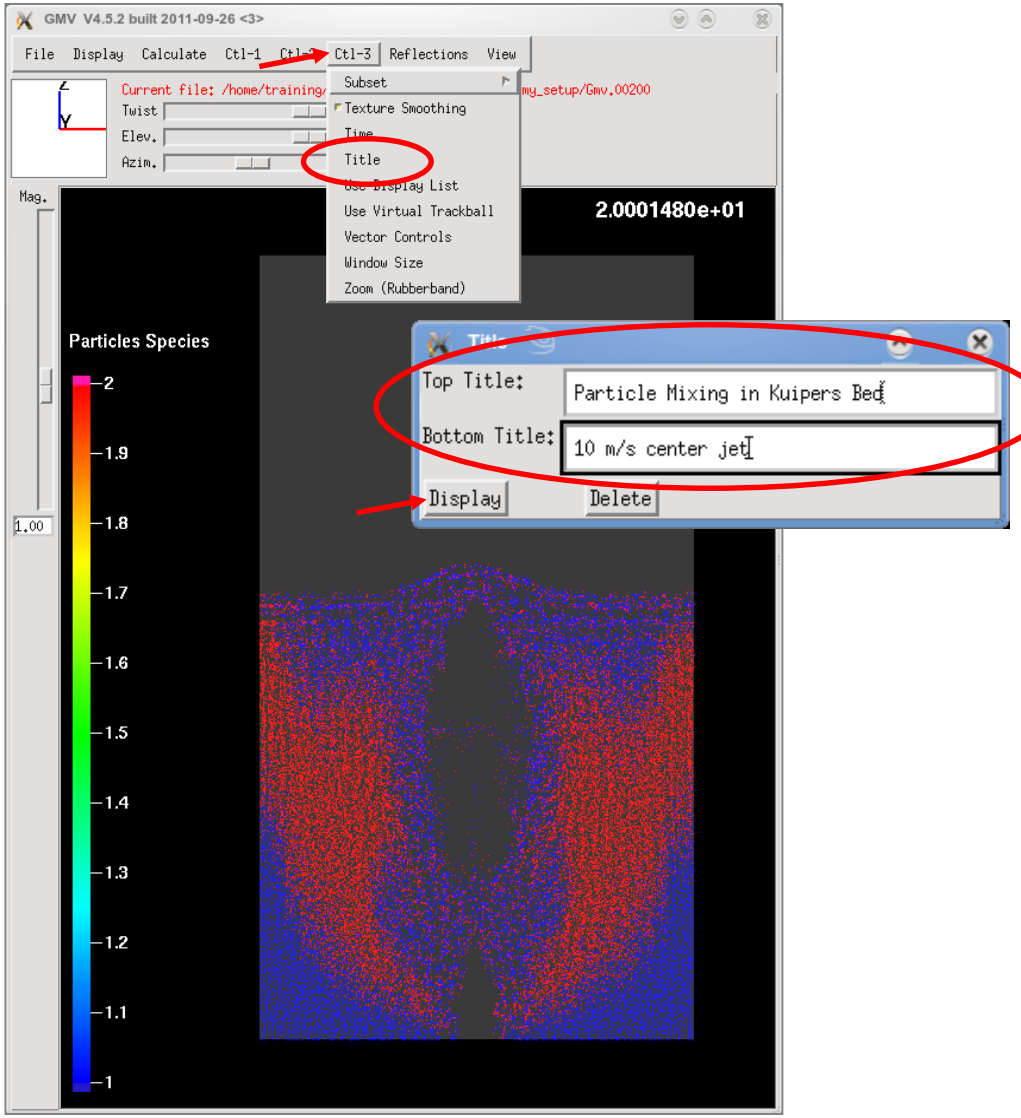
Transparent Geometry

- As a default, the model geometry is displayed in grey (Aluminum) at 30% transparency. Typically, on a black background, it is better to set the transparency to a 10% - 15% range
- Select **Ctl-1 → Coloredit → Materials, Isosurfaces, Isovolumes**
- To see the current cell color:
  Select **Get Color** and then click **Cells**
- Set the **Trans sliders** to a value between 0.10 and 0.15
- To set this as the cell color:
  Select **Set Color** and click on the **Cells** entity to apply the change



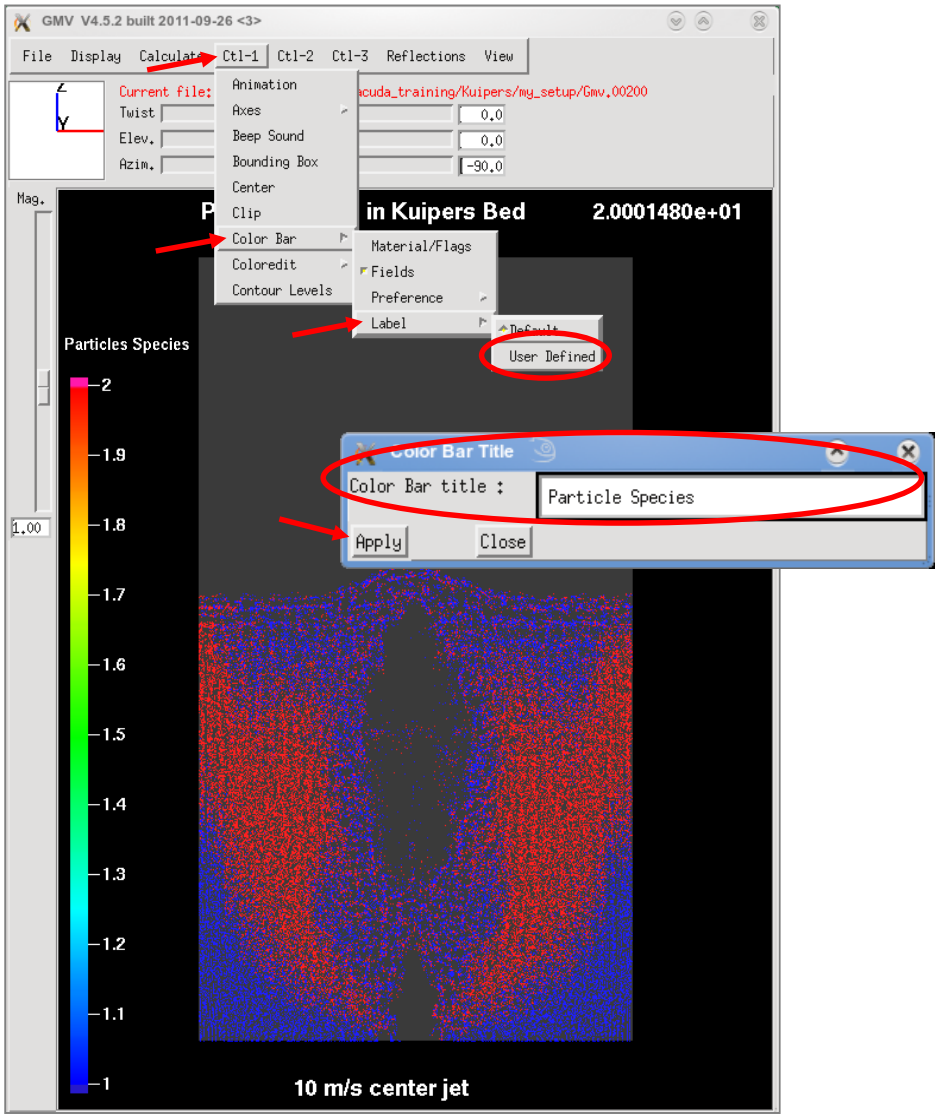
Adding a Title

- Select **Ctl-3 → Title** to raise the Title window
- Enter a top and a bottom title. For example: “Particle Mixing in Kuipers Bed” and “10 m/s center jet”
- Click **Display**



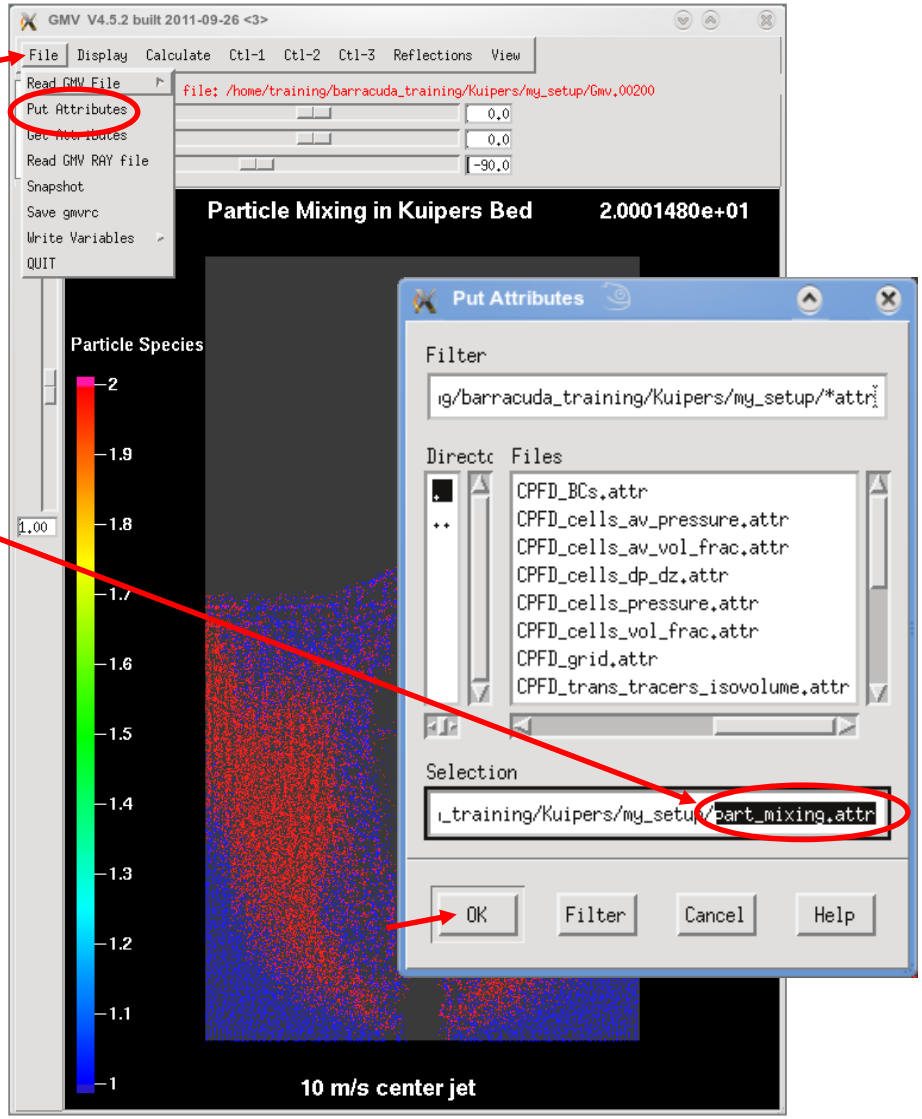
Color Bar Label

- To edit the color bar label: Select **Ctl-1 → Color Bar → Label → User defined**
- Enter a new label. For example: “Particle Species”
- Click **Apply**



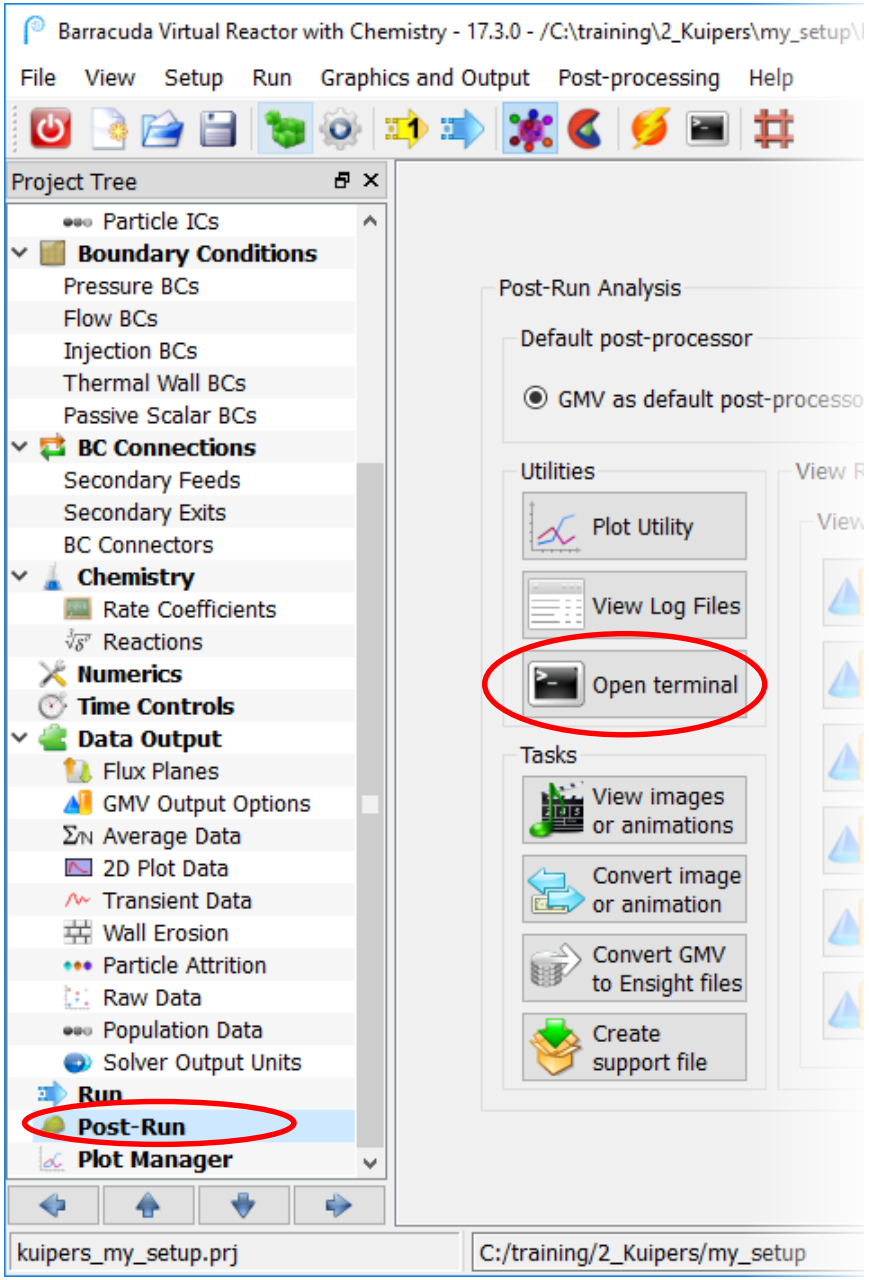
Saving an Attribute File

- When finished setting the data view, it's good practice to save it
- Select **File → Put Attributes** to raise the Put Attributes window
- In the **Selection** box, enter a file name. For example: "part_mixing.attr"
- **NOTE**: Be sure to enter ".attr" at the end of the file name
- Click **OK**. The attribute file can be read at a later time to recreate this specific view by using the **File → Get Attributes** option.



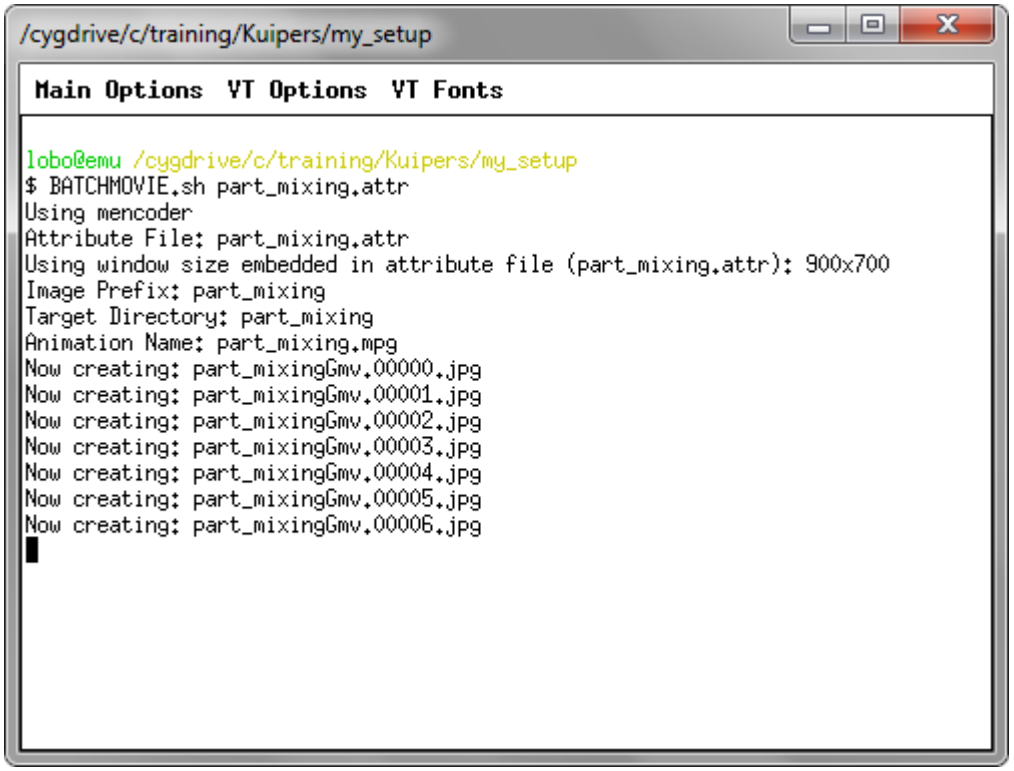
BATCHMOVIE.sh

- Once an attribute file has been saved, you can use BATCHMOVIE.sh to create a movie based on that attribute file.
  - BATCHMOVIE.sh is a script included with Barracuda VR
  - It is used to create movies from the command-line interface
  - It is an efficient tool for creating movies
- In order to use BATCHMOVIE.sh, you must first open a command-line terminal:
- In Windows:
  - Open a Windows CMD terminal by choosing **Post-Run → Open Terminal**
  - In the CMD terminal, type: **xterm**
  - The xterm terminal that opens can be used to run command-line utilities included with Barracuda VR.
  - **Note**: do not close the CMD terminal, because it will also cause the xterm to close.
- In Linux:
  - Open an xterm by choosing **Post-Run → Open Terminal**



Creating a Movie using BATCHMOVIE.sh

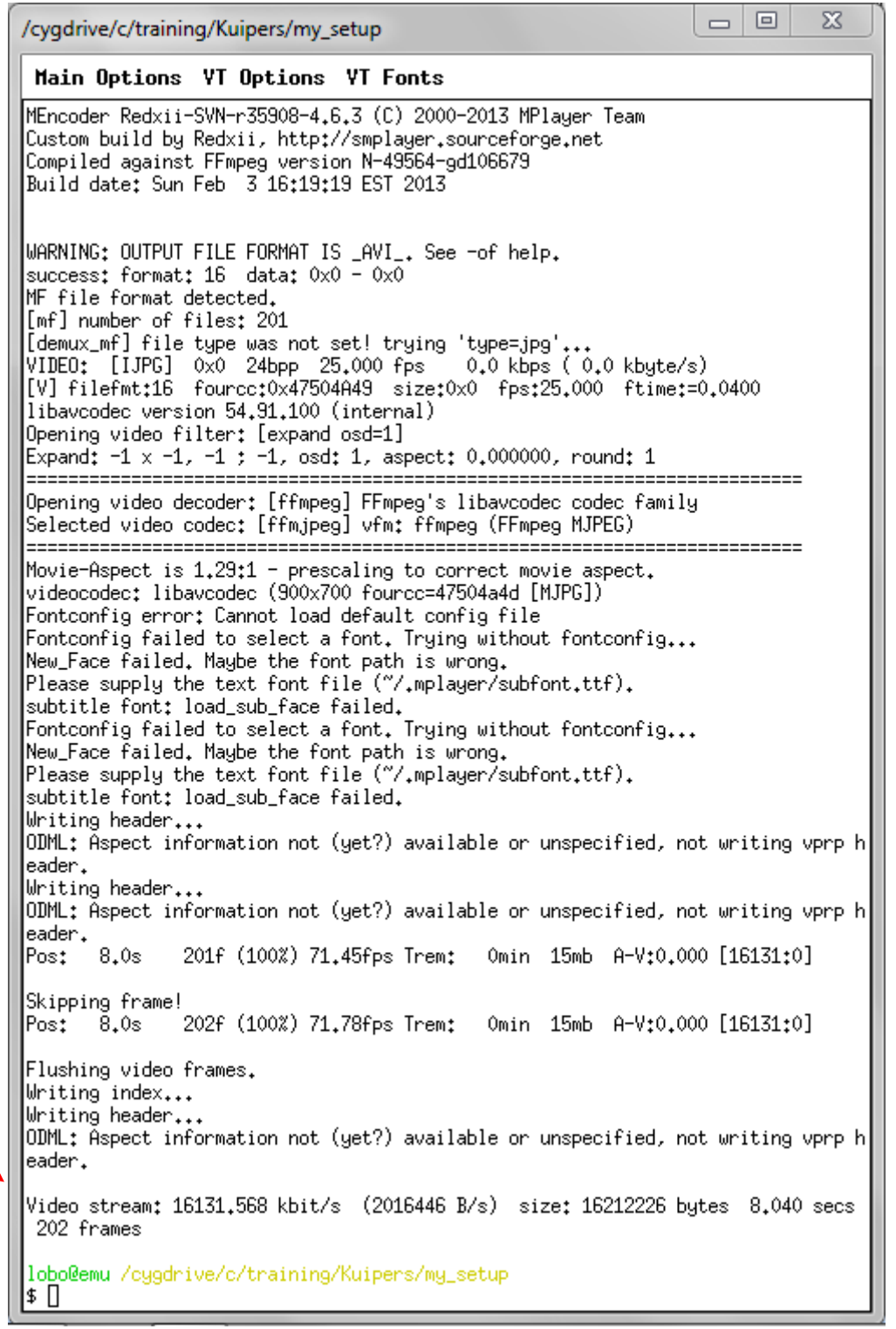
- Continuing the example from above, in which the attribute file was saved as: part_mixing.attr
- In the xterm, type: **`BATCHMOVIE.sh part_mixing.attr`**



- When BATCHMOVIE.sh runs, it creates a sub-directory named after the attribute file, and makes .jpg images within the sub-directory for each Gmv* file from the Barracuda VR simulation.

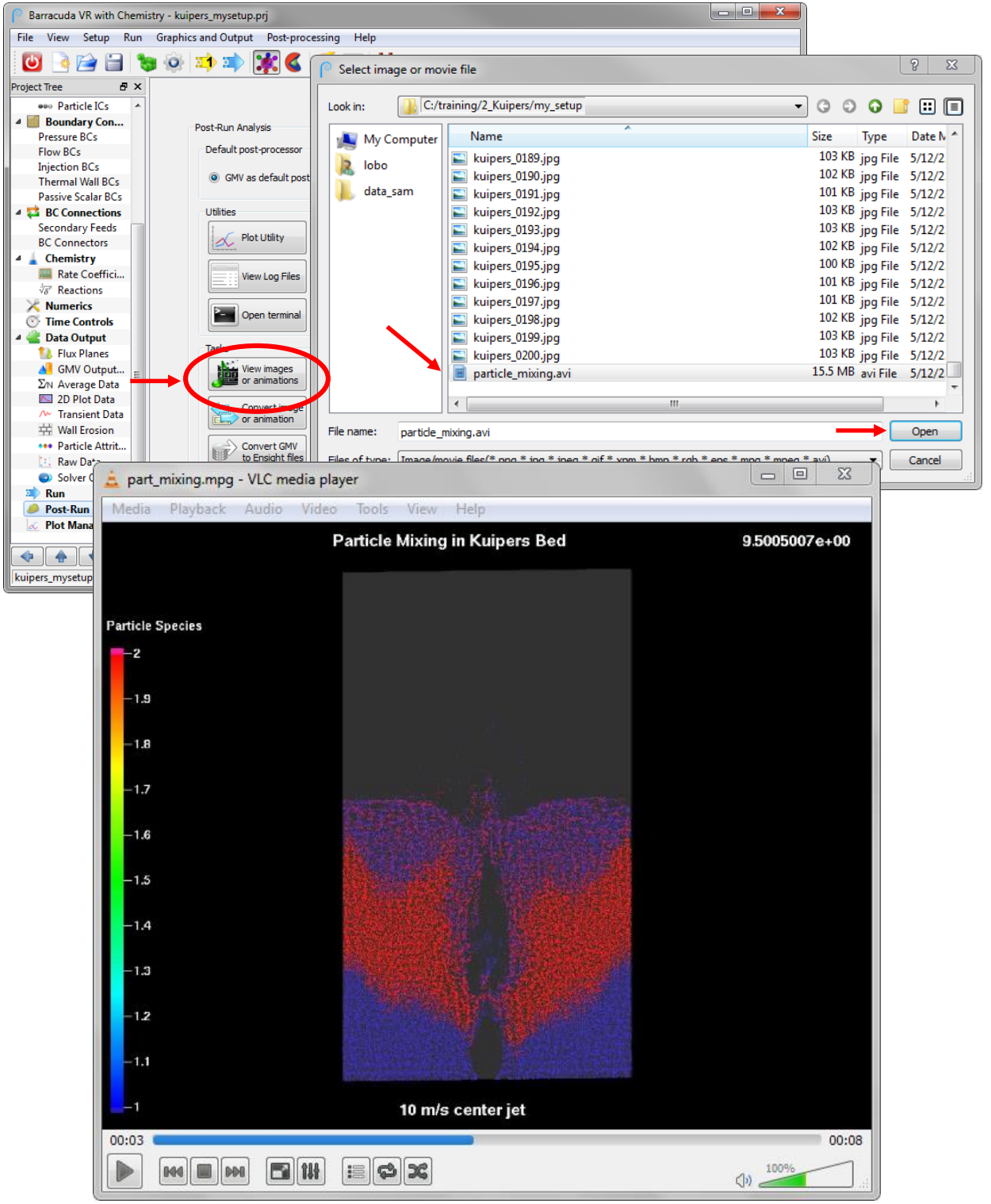
mencoder Output in xterm

- Once .jpg images are created for all Gmv* files, the final step of BATCHMOVIE.sh creates a .mpg movie from the images.
- A software tool called **mencoder** is used to create the final movie.
- This is what the mencoder step looks like when in runs successfully.
- **Note**: many messages are printed to the screen, but normally they are not errors.
- Look at the bottom of the terminal. If there is "Video stream:" information and the command prompt is active, the movie has been successfully created.



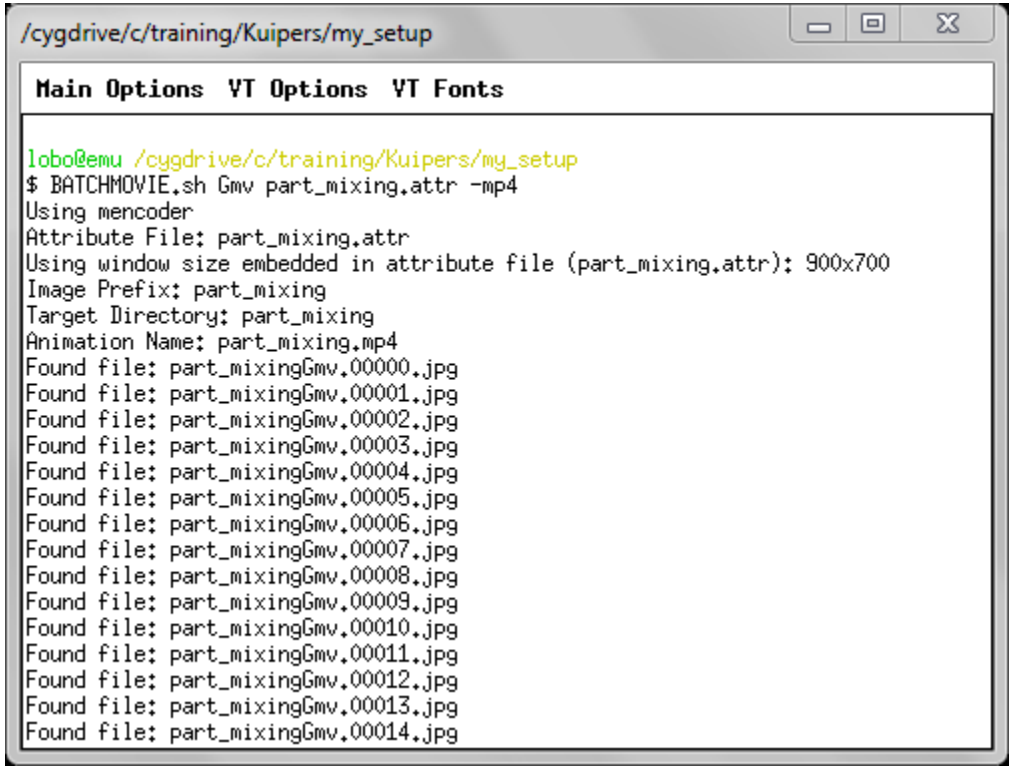
Viewing Animations

- Viewing animations is similar to viewing images. Click on **View images or animations**
- Select the file: particle_mixing.mpg
- Click **Open**
- Barracuda VR will open the animation using your system's default video player
- In Windows:
  - Windows Media Player is usually the default video player. In order to play the .mpg file created by Barracuda VR, it is often necessary to install extra codec packages.
  - VLC is a third-party video player that works very well on windows, and will generally play any movie created by Barracuda VR. If possible, it is recommended that you install VLC and use it as your default video player.
  - Another option is to use the **`-mp4`** option with BATCHMOVIE.sh (see next slide)



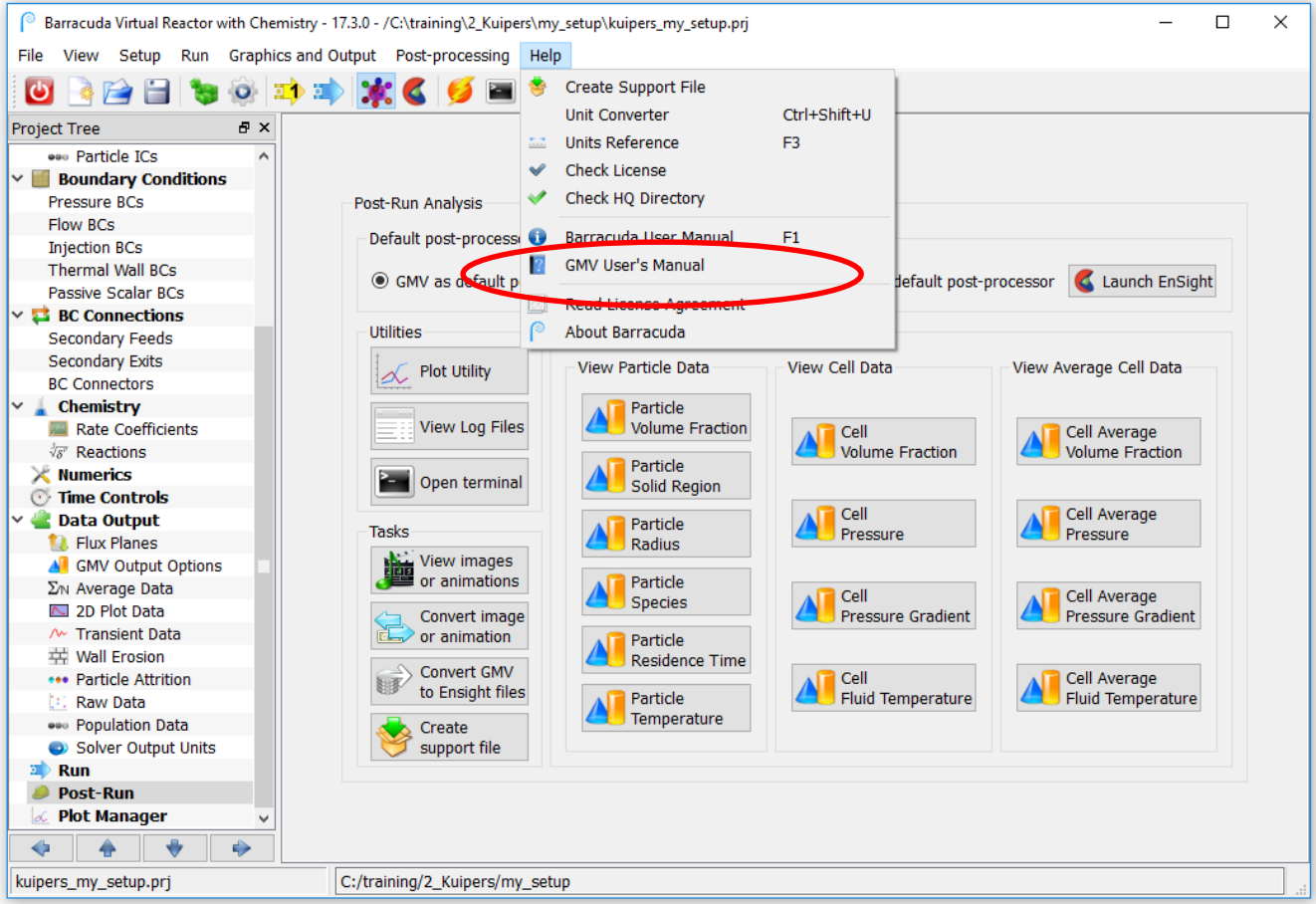
Using the `-mp4` Option with BATCHMOVIE.sh

- If Windows Media Player cannot play the .mpg file that is created by BATCHMOVIE.sh, in most cases using the **`-mp4`** option will generate a movie that can be played.
  - Note: when this option is used, both .mpg and .mp4 format movies are generated.
  - Try playing the .mp4 movie in Windows Media Player.
- Continuing the example from above, in which the attribute file was saved as: part_mixing.attr
- In the xterm, type: **`BATCHMOVIE.sh Gmv part_mixing.attr -mp4`**



GMV User's Manual

- For more detailed information on GMV features, check the **GMV User's Manual** under the **Help** button

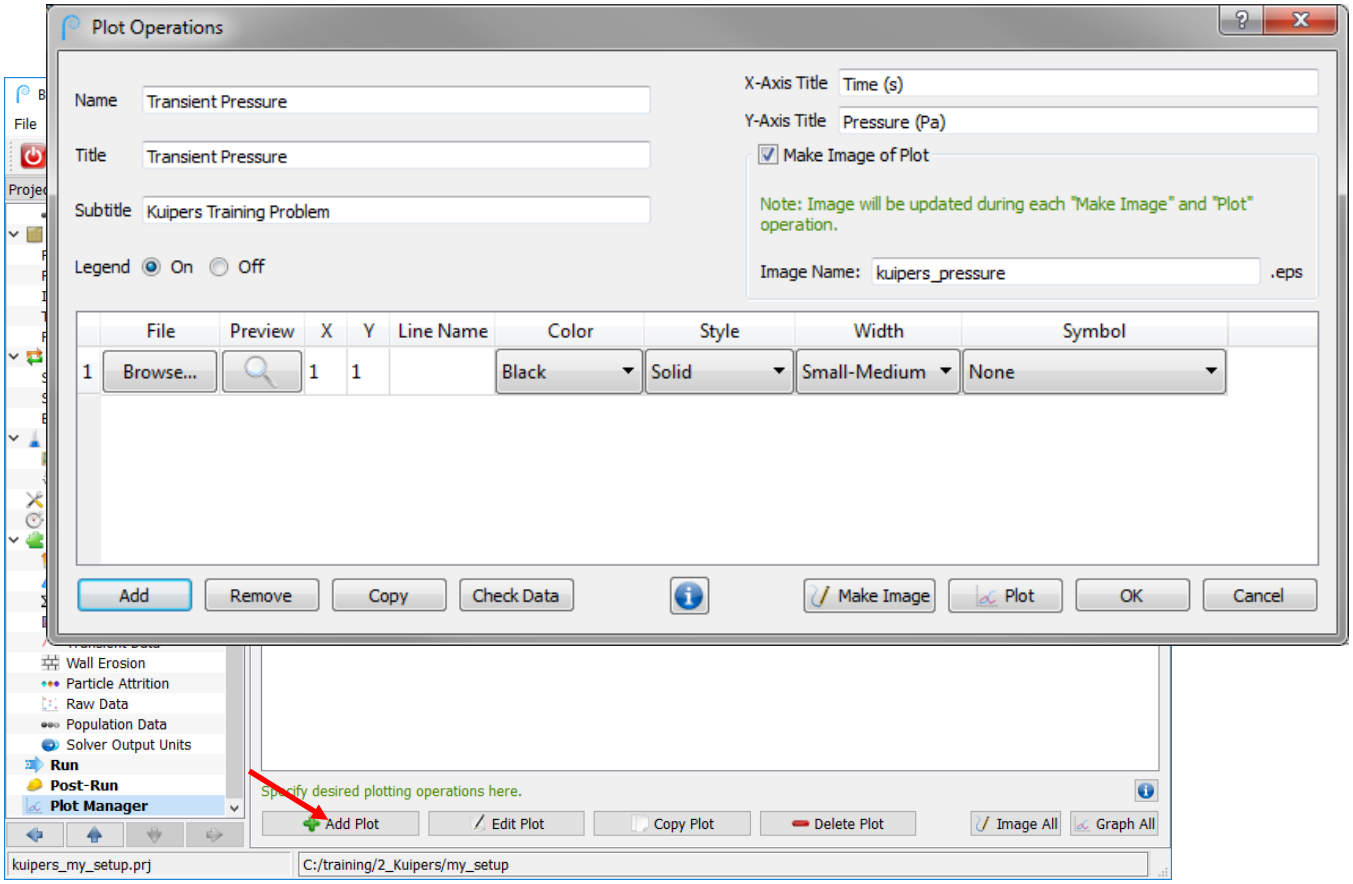


GMV Basics Recap

- We have now covered the basics of how to view common simulation results and create high quality images and animations in GMV
- Viewing Gmv files is not the only way simulation results can be analyzed
- Next, 2D plotting of data in Barracuda will be covered

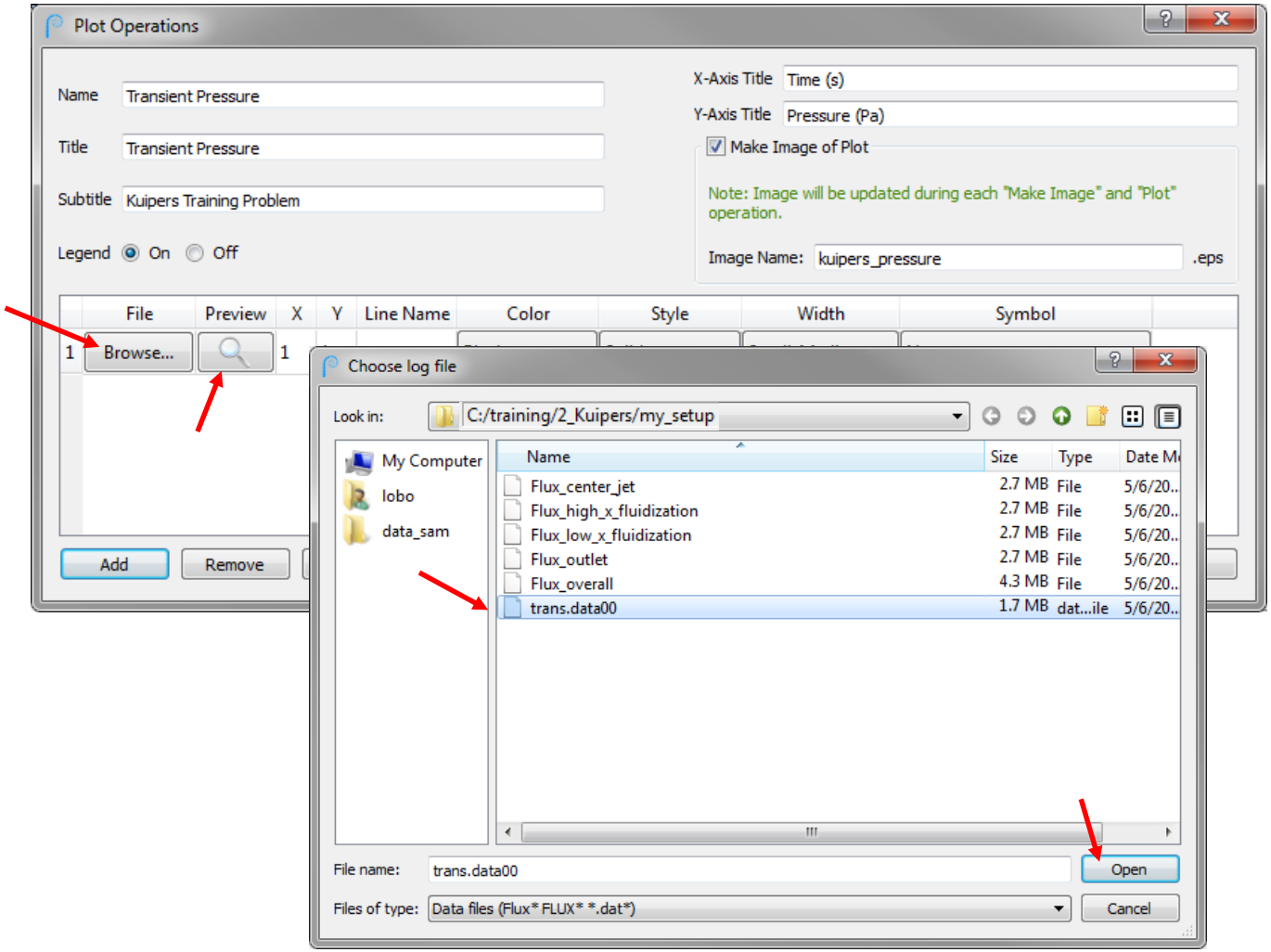
Plotting Transient Data

- When setting up the simulation, we requested that transient data for the pressure at specific cells in the Kuipers bed be collected
- That data can now be plotted in **Plot Manager**
- Click **Add Plot**
- Add a **Name**, **Title**, **Subtitle**, **X-Axis Title**, and **Y-Axis Title**
- Select **Make Image of Plot** and give an appropriate **Image Name**



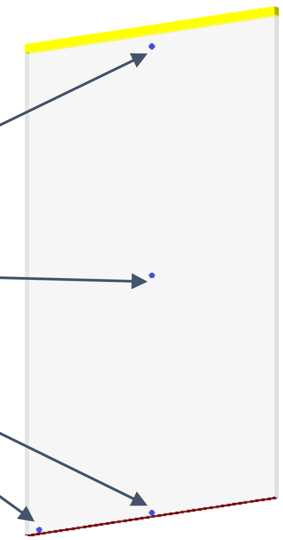
Viewing Log Files

- To choose the file containing data to plot, click on **Browse**
- Select the file: trans.data00
- Click **Open**
- Then click **Preview** to see the log files



Transient Data File Contents

- The file **trans.data00** shows the data collected in columns
  - Column 1 corresponds to Time
  - Column 2 corresponds to Pressure at cell (2, 1, 1)
  - Column 3 corresponds to Pressure at cell (16, 1, 1)
  - Column 4 corresponds to Pressure at cell (16, 1, 30)
  - Column 5 corresponds to Pressure at cell (16, 1, 58)



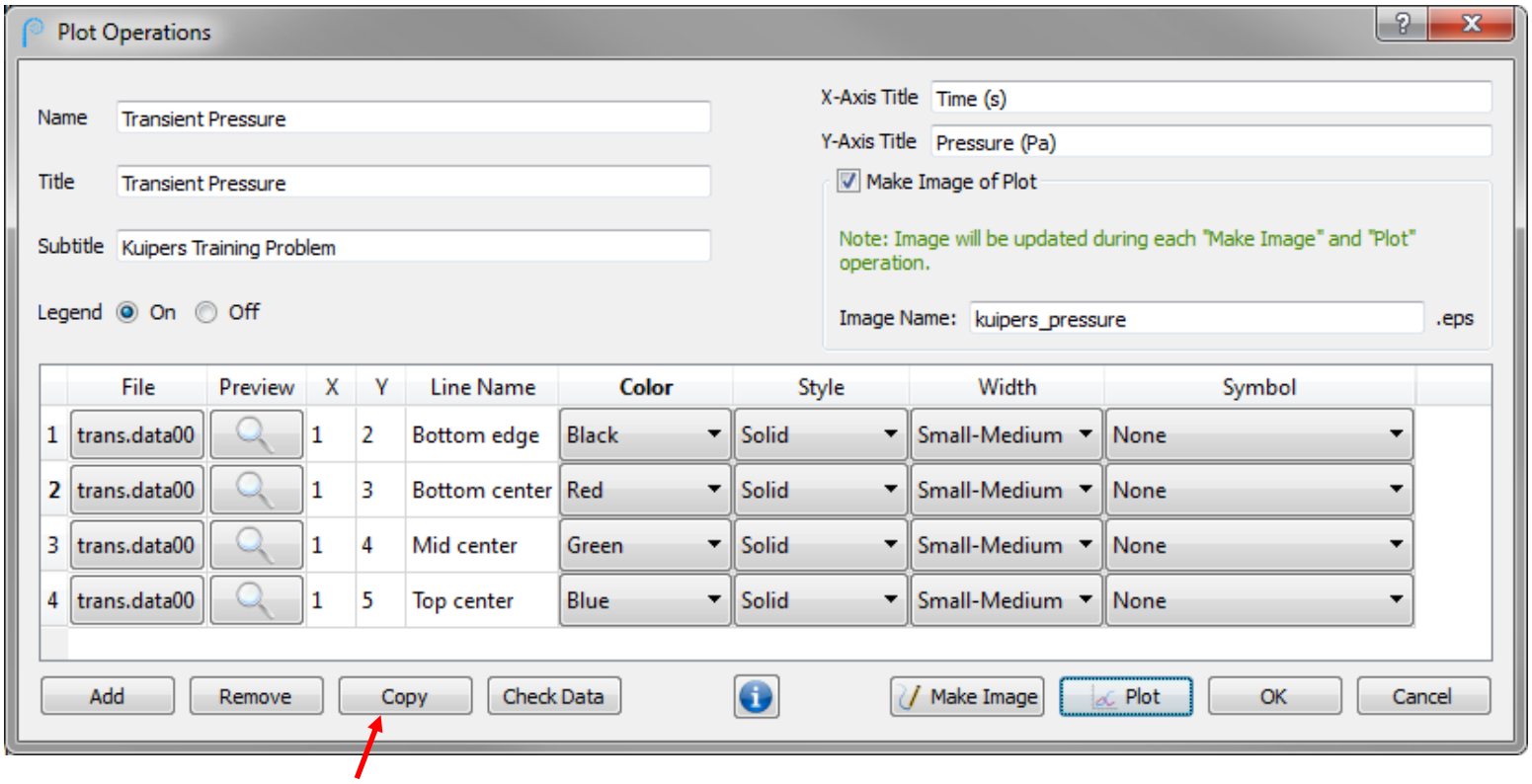
Preview

```
#Wed May 06 10:56:20 2015
#
# Barracuda release 17.0.0.
# Solver version 17.0.0.x153. Build date Wed Apr 22 15:31:36 MDT 2015.
# Compiled with x86_64
#
#       Variable name                                          Units                     ijk            xyz(m)                                    Comment
#       -----------------------------------------------------  ------------------------  ------------  ----------------------------------------  ----------
#@   1  "Time"                                                 "s"
#@   2  "Pressure"                                             "Pa"                      "  2   1   1"  "2.57250e-01  7.50000e-03  8.33333e-03"  " "
#@   3  "Pressure"                                             "Pa"                      " 16   1   1"  "0.00000e+00  7.50000e-03  8.33333e-03"  " "
#@   4  "Pressure"                                             "Pa"                      " 16   1  30"  "0.00000e+00  7.50000e-03  4.91667e-01"  " "
#@   5  "Pressure"                                             "Pa"                      " 16   1  58"  "0.00000e+00  7.50000e-03  9.58333e-01"  " "
```

Show Entire File   OK

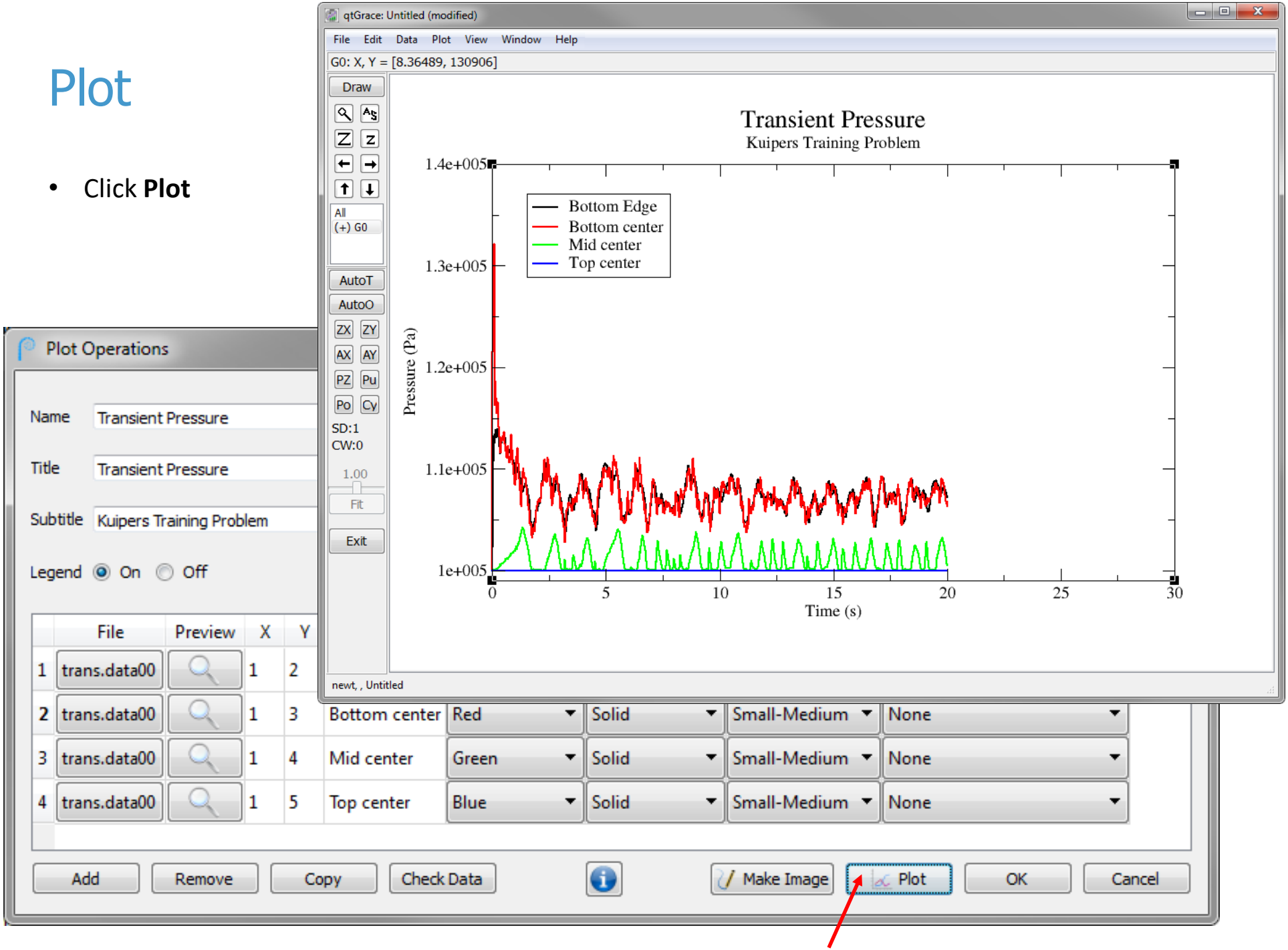
Selecting Plot Data

- In the Plot Operations dialog, select the data you wish to plot:
  - Since column 1 in the trans.data00 file corresponds to time, **X** will remain as "1".
  - Highlight the first row and click **Copy** 3 times. This gives you the necessary number of rows for your plot.
  - Set **Y** to "2". Remember this is the pressure at node (2, 1, 1). Then set **Y** to "3", "4", and "5" in the subsequent rows.
  - Add a **Line Name** for each row, which will correspond to the legend text for each line on the plot
  - Set the **Color** differently for each row



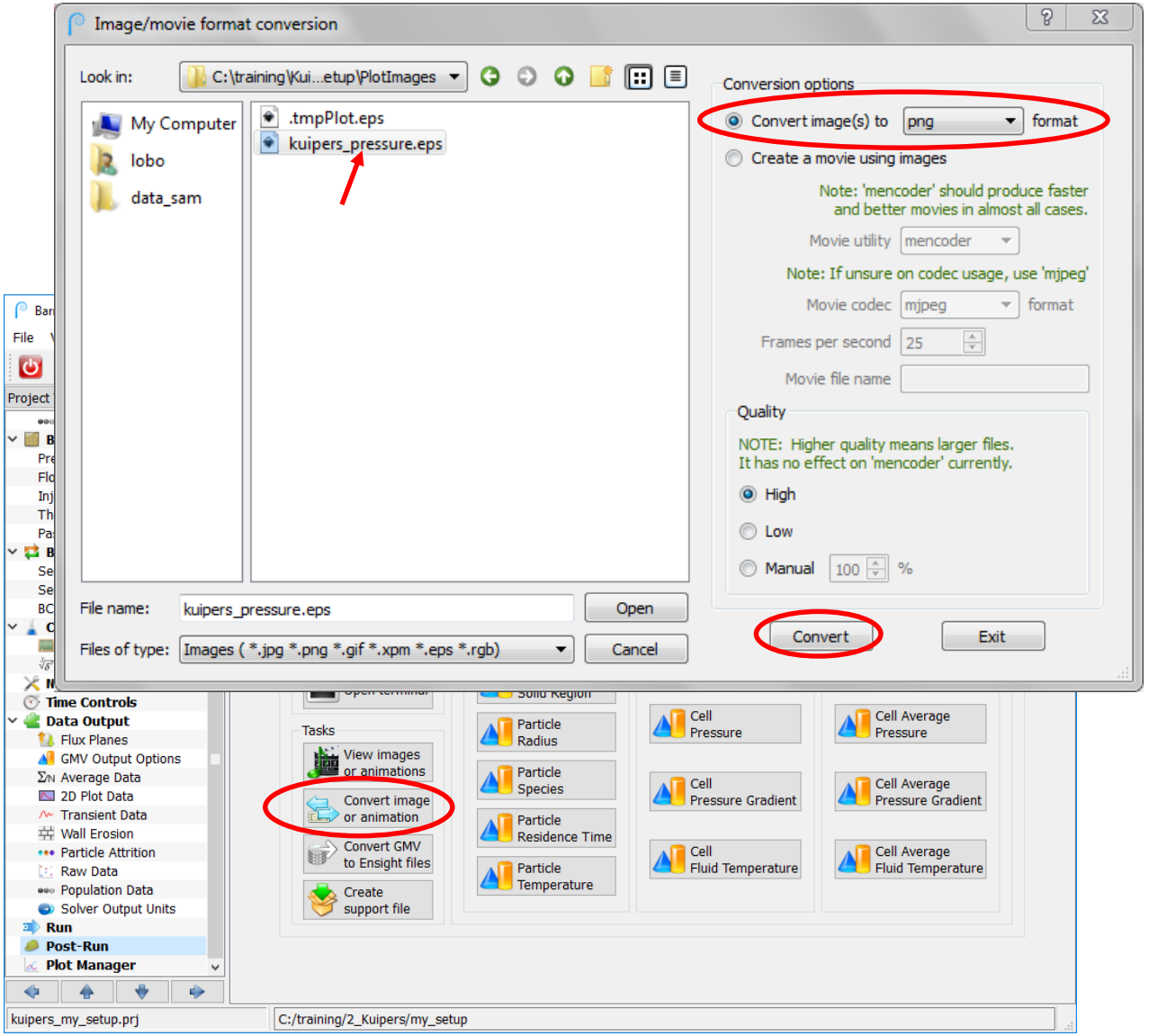
Plot

- Click **Plot**



Converting an .eps Image to .png Format

- By default, images created by **Plot Manager** are in .eps (Encapsulated Post-Script) format.
  - This format is good for publication-quality plots, because it is a vector format that stays high-quality even if its size is changed.
  - However, .eps is not very common, and often it is better to share images in a different format, such as .png.
- To convert an .eps file to a .png image, go to **Post-Run** and click on the **Convert image or animation** button
- Select the file you wish to convert
  - **Note**: Plot Manager creates a sub-folder called **PlotImages** and stores all plot images in this sub-folder.
- Select **Convert image(s) to** and **png** format
- Choose the **Quality** of the image
- Click on **Convert**



Plot Utility Reference Materials

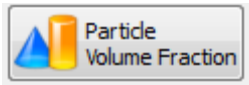
- On Linux, the plot utility is Xmgr. More information on XMGR features is available online:
  - Xmgr home page: http://plasma-gate.weizmann.ac.il/Xmgr/
  - Xmgr User Guide: http://plasma-gate.weizmann.ac.il/Xmgr/doc/xmgr.html
- On Windows, the plot utility is QtGrace. More information on Grace is available online:
  - Grace home page: http://plasma-gate.weizmann.ac.il/Grace/
  - Grace User Guide: http://plasma-gate.weizmann.ac.il/Grace/

Part II: Advanced Post-Processing

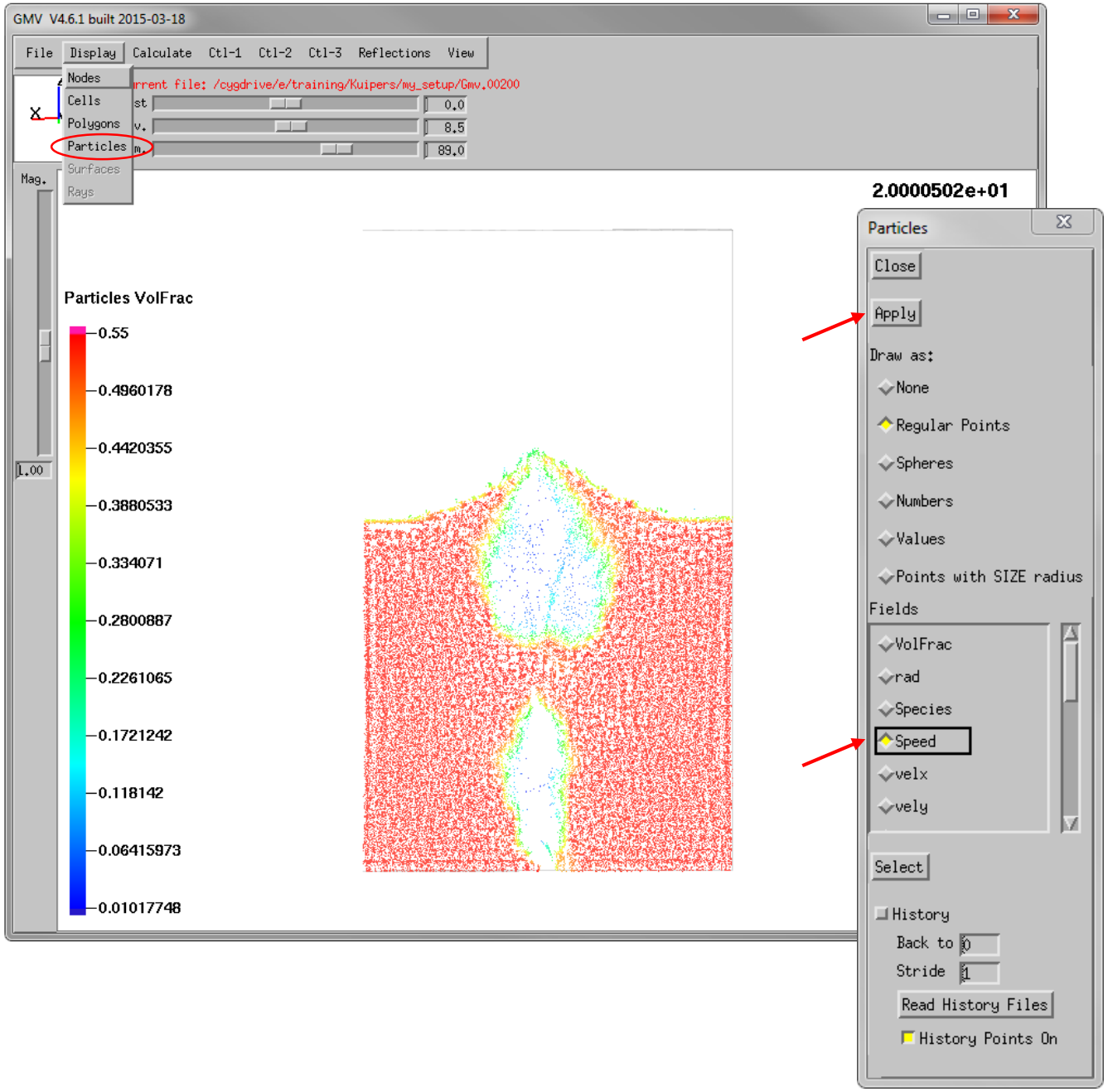
- Changing data limits
- Building vectors (arrow plot)
  - Instantaneous Velocity
  - Average Velocity
- Changing vector view settings
- Creating Isovolume views

Particles

- When a field of data is needed that does not have a specific GMV button on the **Post Run** page, there are ways to utilize the shortcut buttons to create the views needed
- First, determine if the view needed is based on particles or cells
- For our example, we are going to concentrate on particle speed
- Click on Particle Volume Fraction

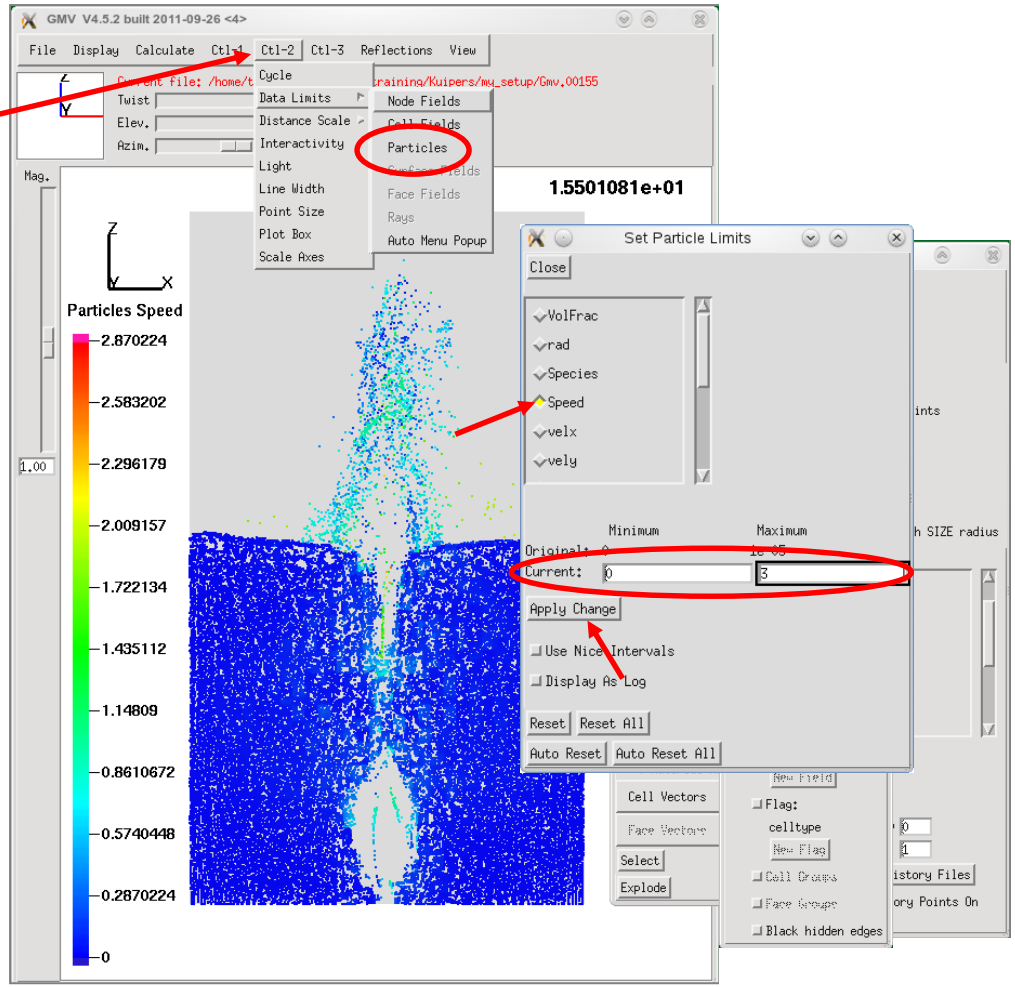


- Click on Display→Particles
- Select Speed as the new Field
- Click Apply



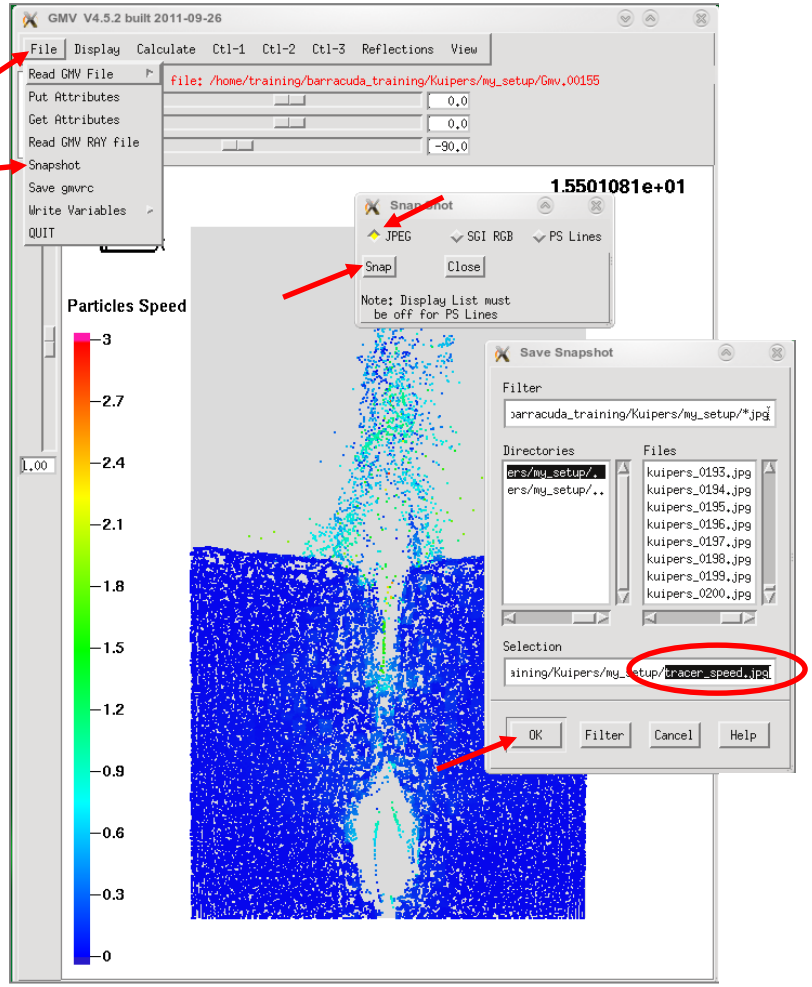
Data Limits

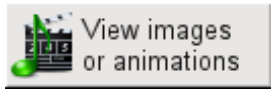
- Set the data limits of the particles speed
- Select **Ctl-2 → Data Limits → Particles**
- A new window will pop up. Set the **Speed** data limits from "0" to "3"
- Click on **Apply Change**
- Notice the change in the color scale values, which correspond to the limits just set.



Taking a Single Snapshot

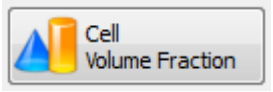
- The current view can be saved as a jpeg image by taking a snapshot in GMV
- Select **File → Snapshot** to raise the snapshot window
- Select **JPEG**, click **Snap**
- Enter a name for the image; for example: “tracer_speed.jpg”
  - Remember to add the .jpg extension
- Click **OK**
- To view the image, in the Barracuda VR GUI, click on **Post-Run → View images or animations** and select the desired file



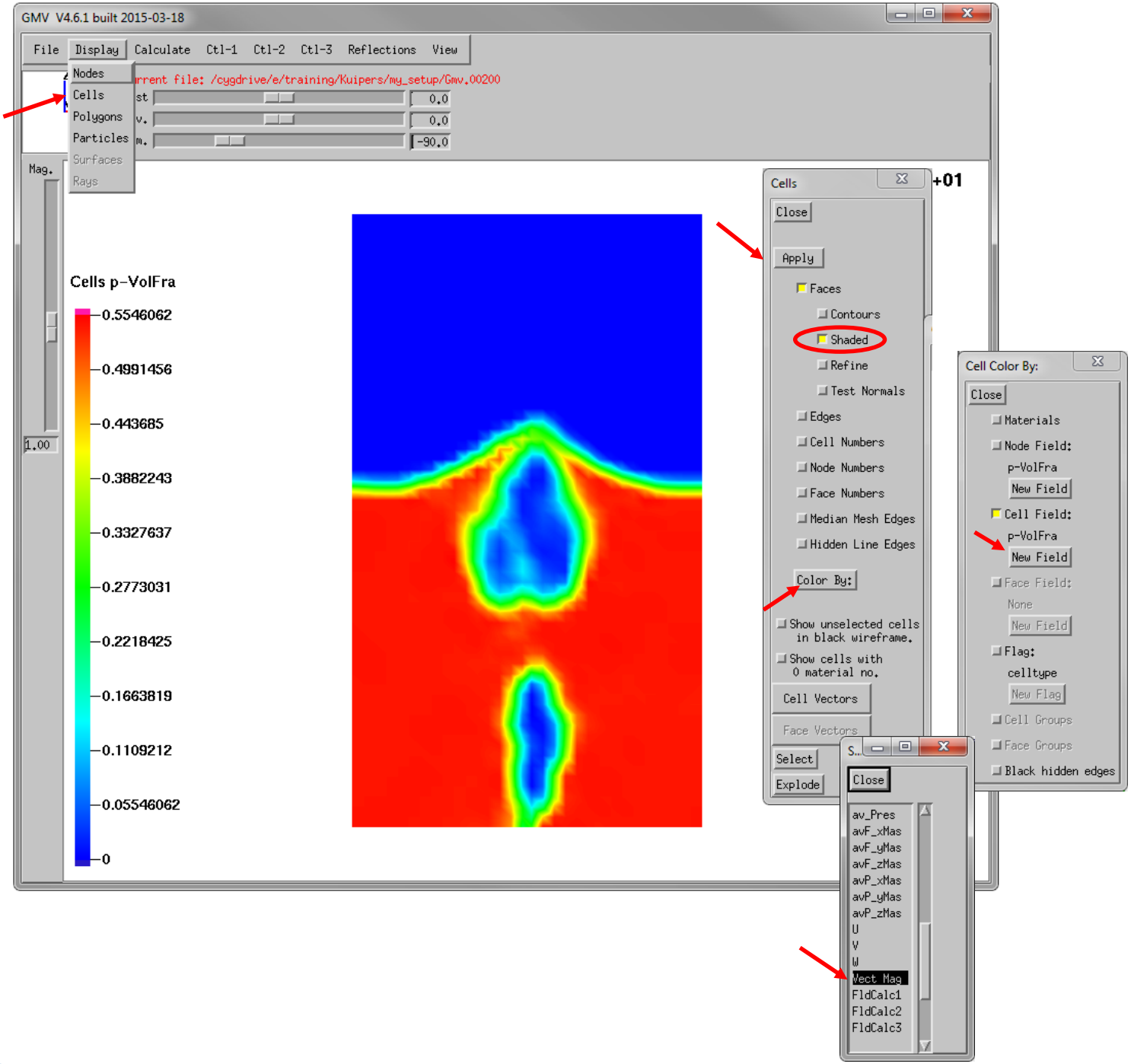


Vectors

- Click on Cell Volume Fraction



- Select **Display → Cells → Color By: → Cell Field → New Field → Vect Mag**
- Make sure to select **Shaded** in the Cells window and click **Apply**

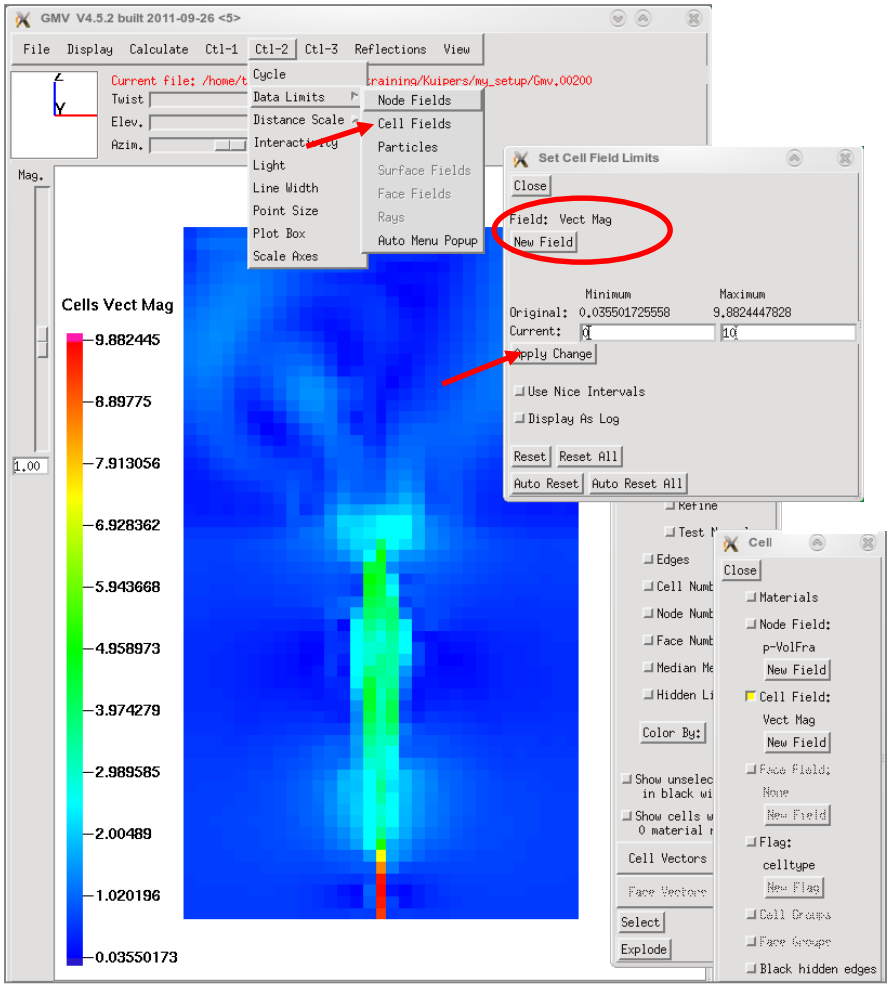


Vector Data Limits

- Pick nice data limits for the vector magnitude

- How do we do this?
  - Select **Ctl-2 → Data Limits → Cell Fields**
  - Select **New Field → Vect Mag**
  - Set new data limits: “0-10”
  - Click **Apply Change**

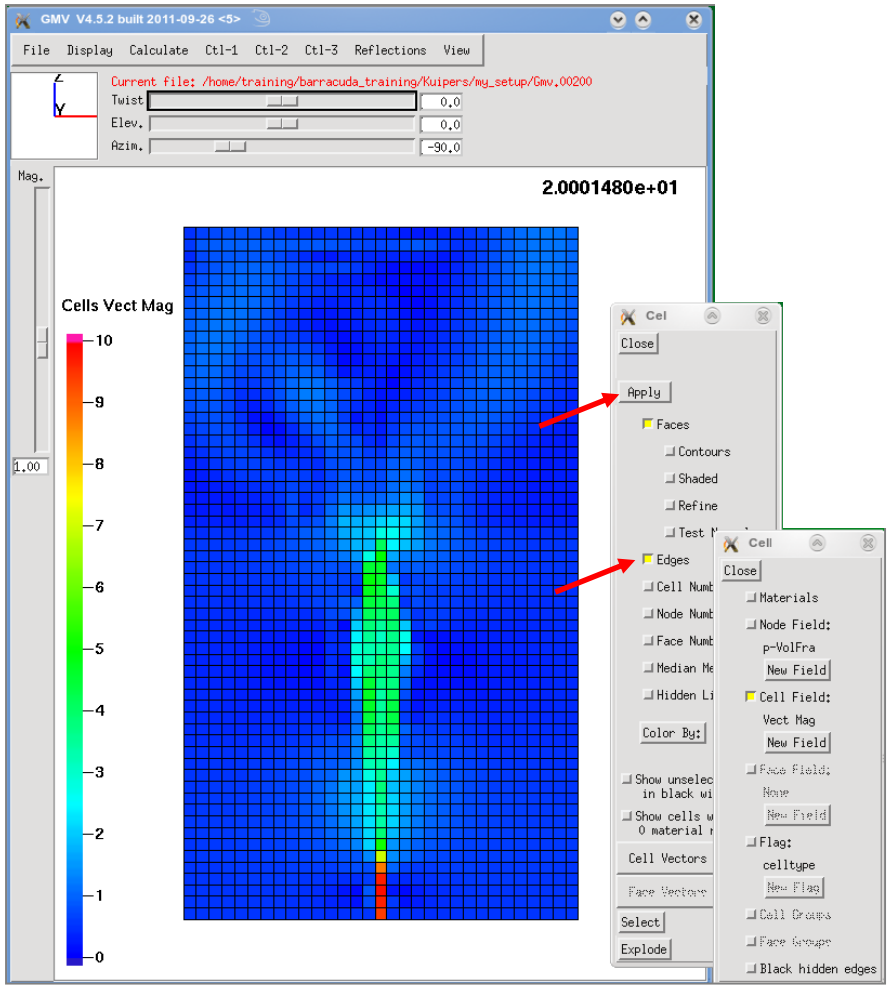
- Question: What does the vector magnitude indicate?



Instantaneous Fluid Velocity

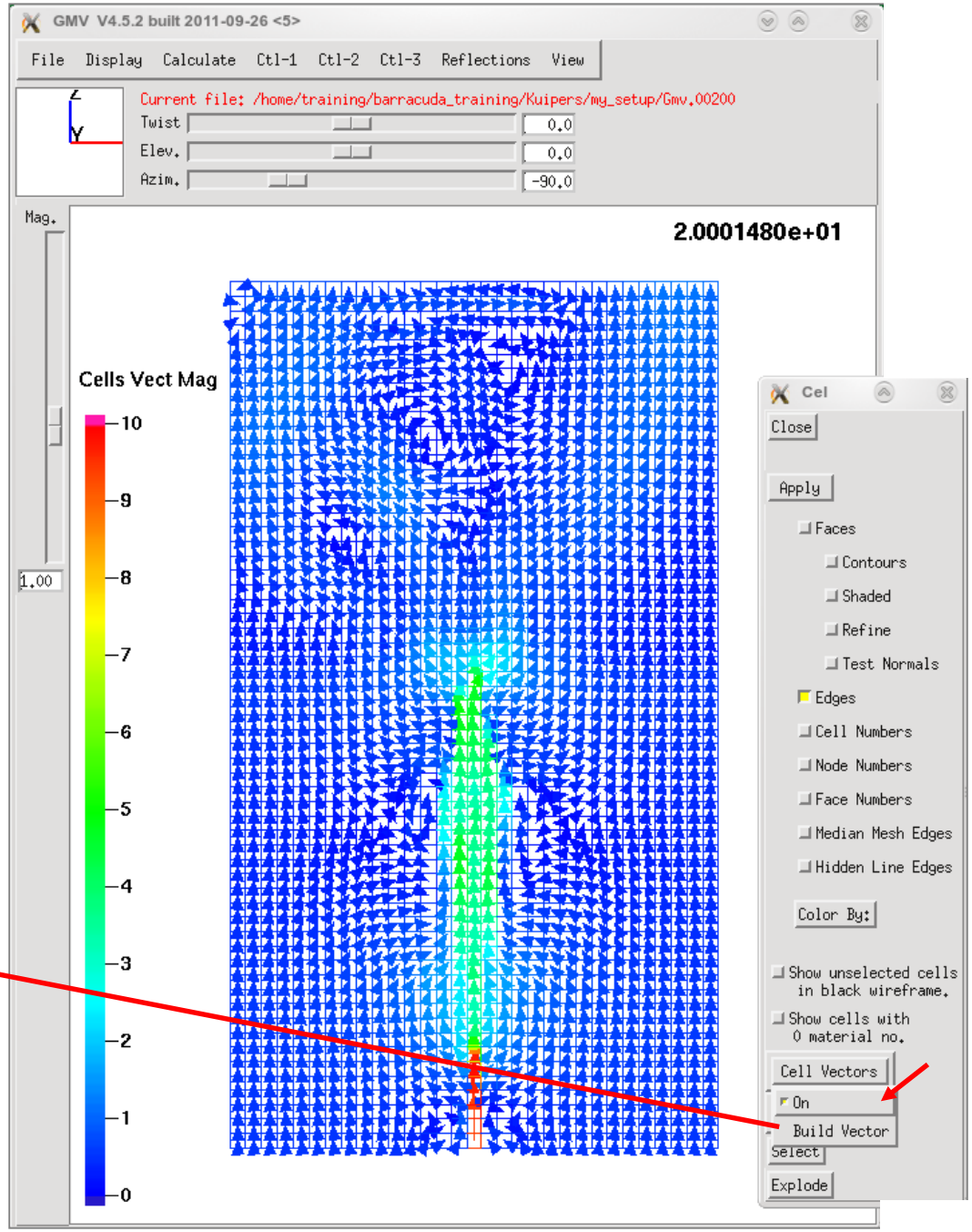
- By default, the vector magnitude indicates the instantaneous fluid velocity

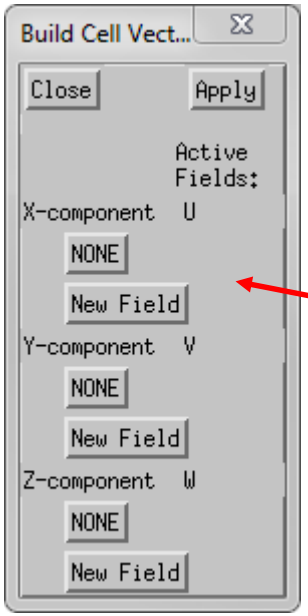
- Now display the grid lines (how?)
  - Select **Edges** in the Cells window
  - Click **Apply**



Displaying Cell Vectors

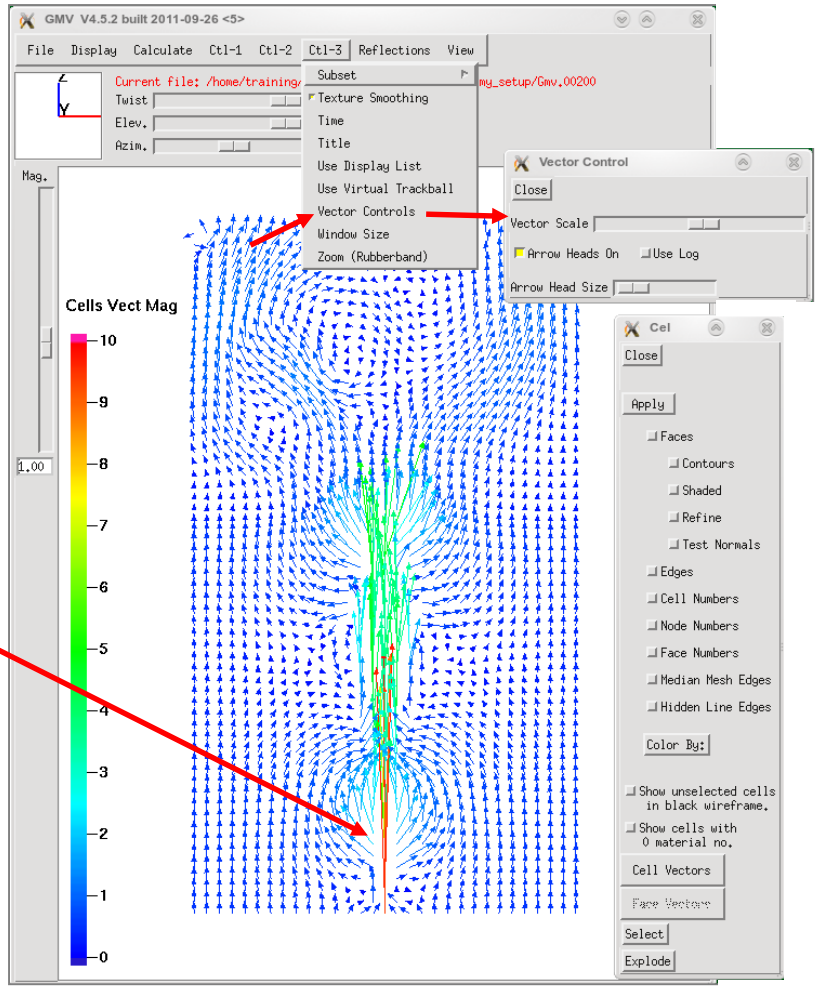
- Turn on Cell Vectors by clicking on **Cells Vectors** in the Cells window
- Select **On**
- Select **Build Vector** and click **Apply**
- Click **Apply** in Cells window
- **Note**: Make sure **Faces** isn't selected in the Cells window. Otherwise the vectors will be blocked from view by the cells





Vector Controls

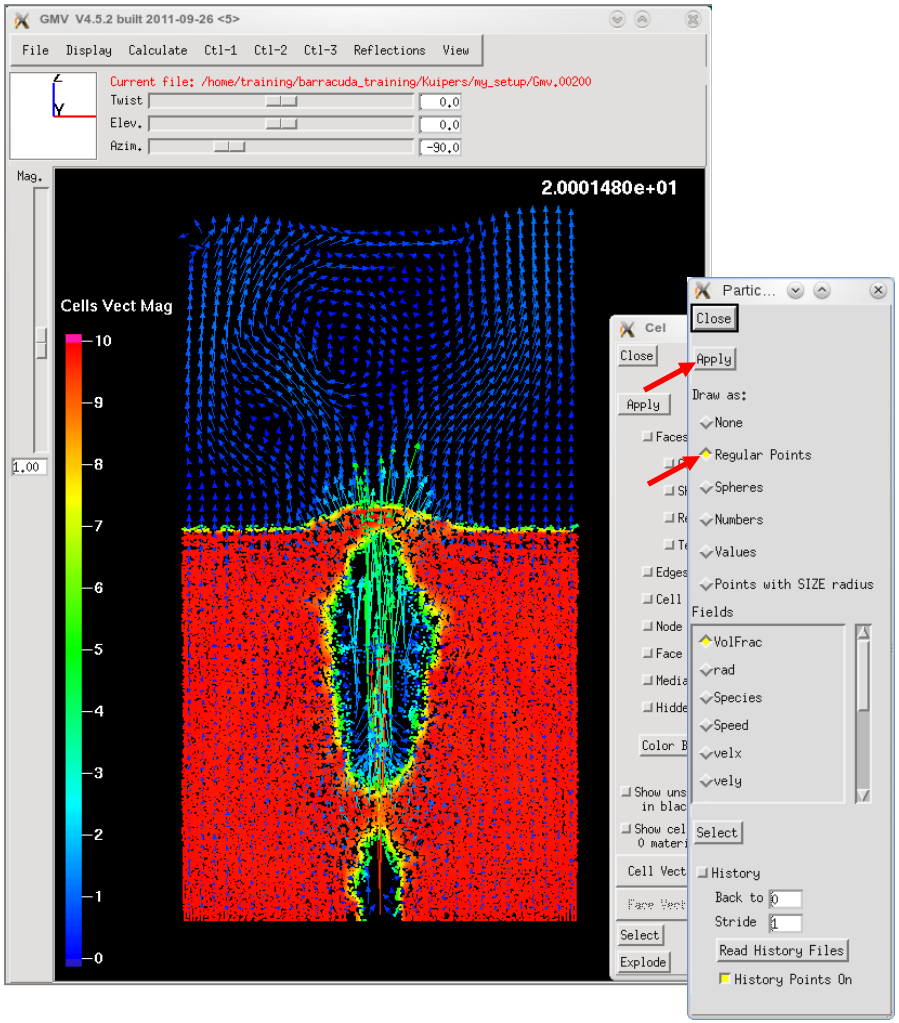
- Remove the grid lines by deselecting **Edges** in the Cell window. Remember to click **Apply**
- You can control the vectors by selecting **Vector Controls** from **Ctl-3**
- You can change the arrow head and vector sizes
- Question: What's happening here?



Displaying Vectors and Particles

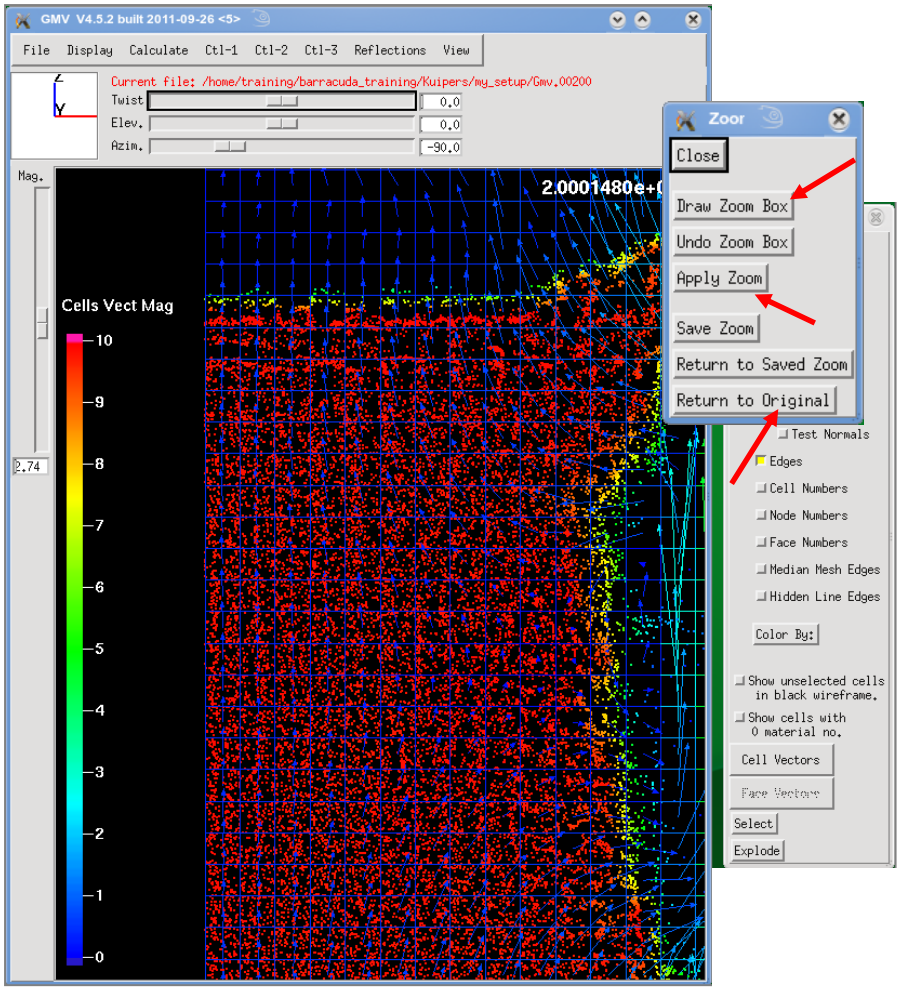
- To better display the vectors, set the background color to black

- Next, superimpose the particles by displaying particles
  - Select **Particles** from the **Display** menu
  - Select **Regular Points** in the Particles window
  - Click **Apply**



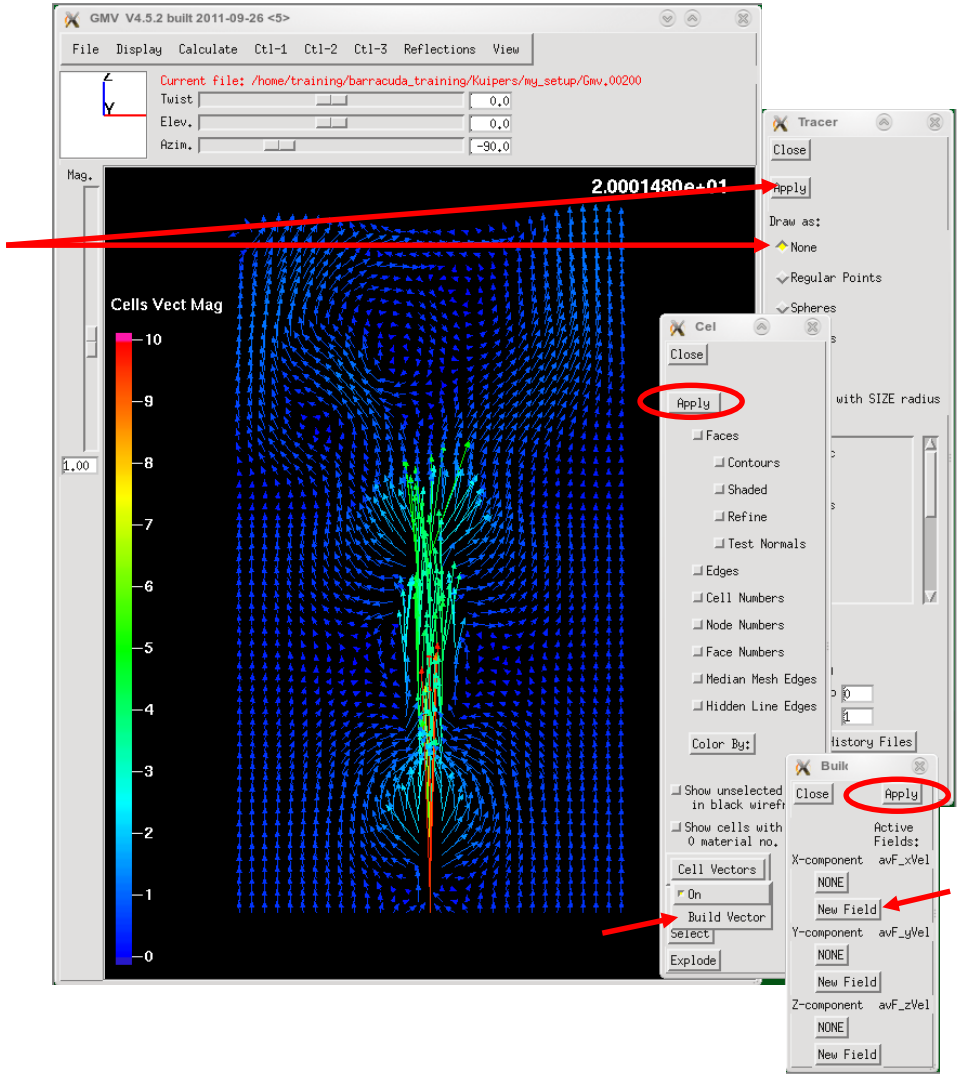
Zooming into Areas of Interest

- To zoom into specific areas of interest, raise the **Zoom** window:
  - Click on **Ctl-3 → Zoom (Rubberband)**
- Click on **Draw Zoom Box** and use the mouse to select the area of the model you'd like to magnify
- Click on **Apply Zoom**
- To reset the view, click on **Return to Original**



Average Fluid Velocity

- Next, create a plot of the average fluid velocity vector
- First remove the particles by selecting **None** in the Particles window. Don't forget to click **Apply**
- Next, click on **Cell Vectors → Build Vector** to raise the Build Cell Vector window
  - Under **X-component**, click on **New Field** and select **avF_xVel** from the list
  - Under **Y-component**, click on **New Field** and select **avF_yVel** from the list
  - Under **Z-component**, click on **New Field** and select **avF_zVel** from the list
- Click **Apply** in first the Build Cell Vector window and then in the Cells window



Vectors

- Question: How does the average fluid velocity at 20 seconds compare with the average fluid velocity at 10 seconds?
  - Has the system reached a steady state at 10 seconds?

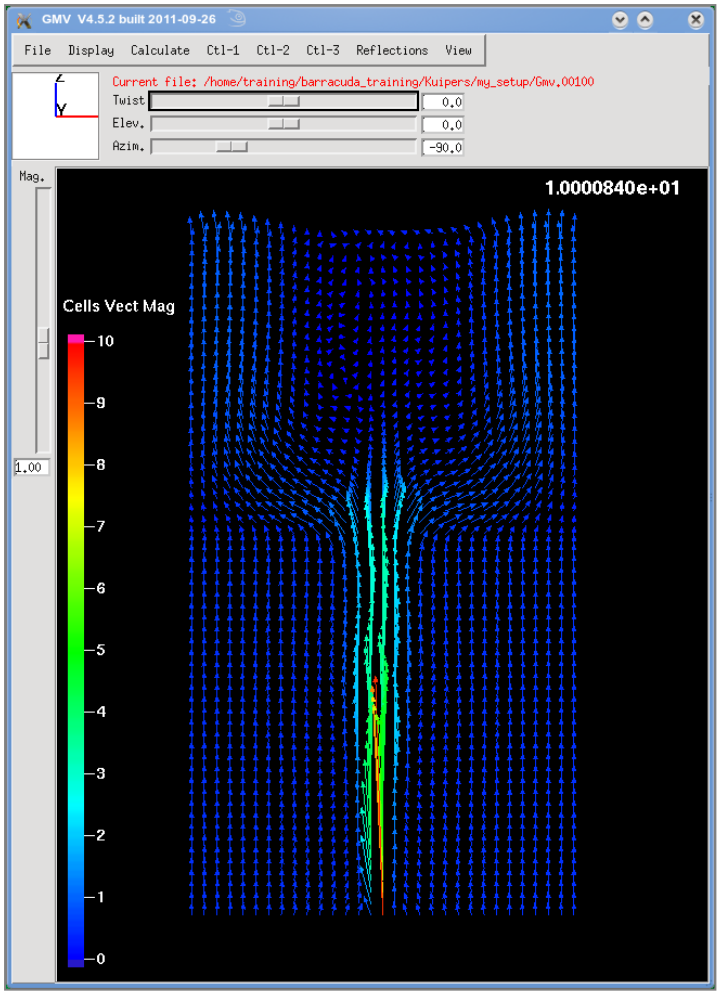
    _____

    _____

  - Is 10 seconds a good average? How can you tell?

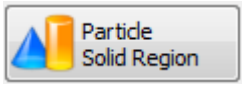
    _____

    _____

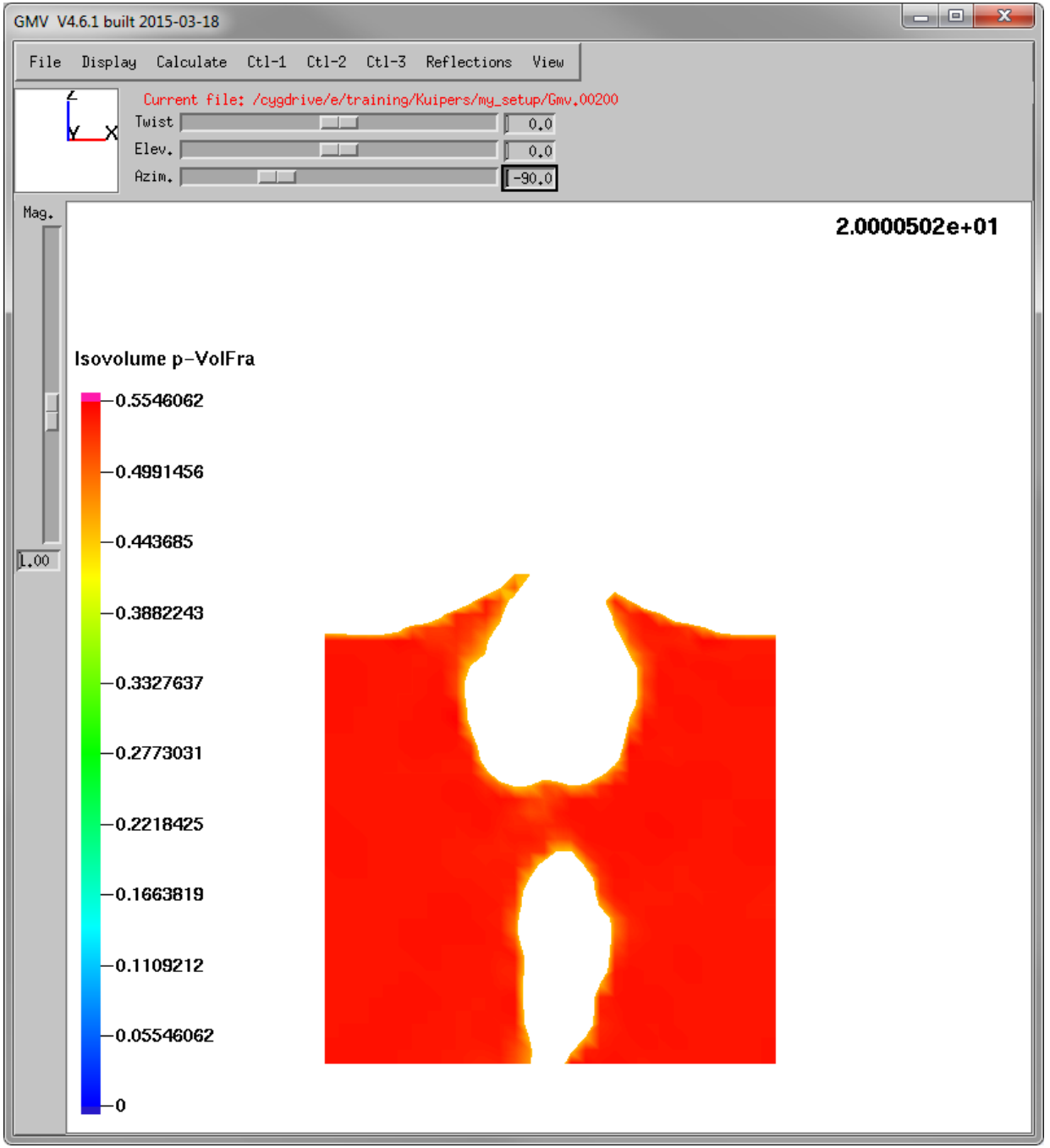


Isovolumes

- It is often nice to display regions of particles which are near close pack as solids
- The Isovolumes feature can help identify the location and shape of air bubbles in the Kuipers bed
- Click on Particle Solid Region

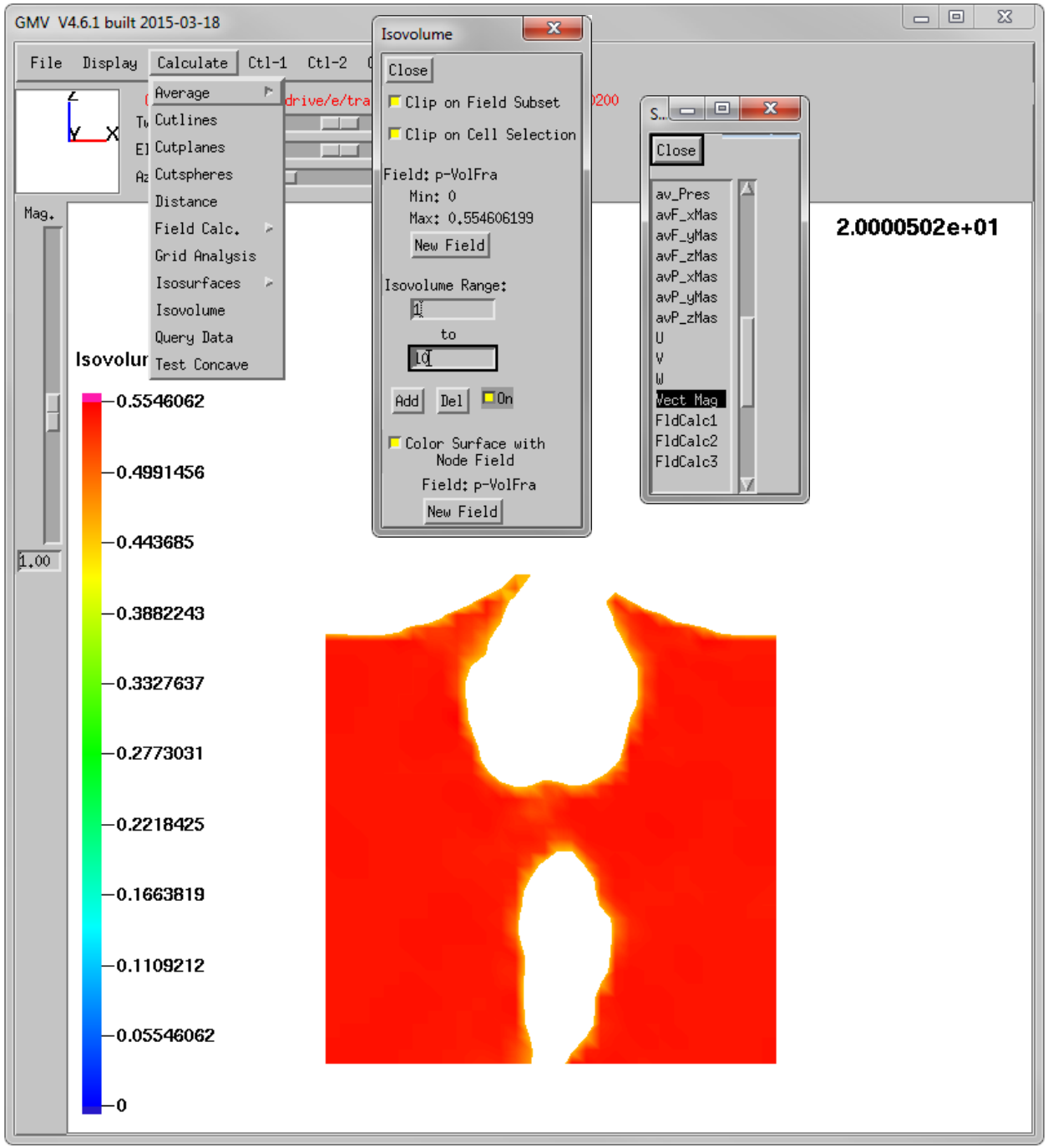


- The Isovolume is colored by particle volume fraction
- Other fields can be specified in order to see different regions displayed as solids, rather than individual particles



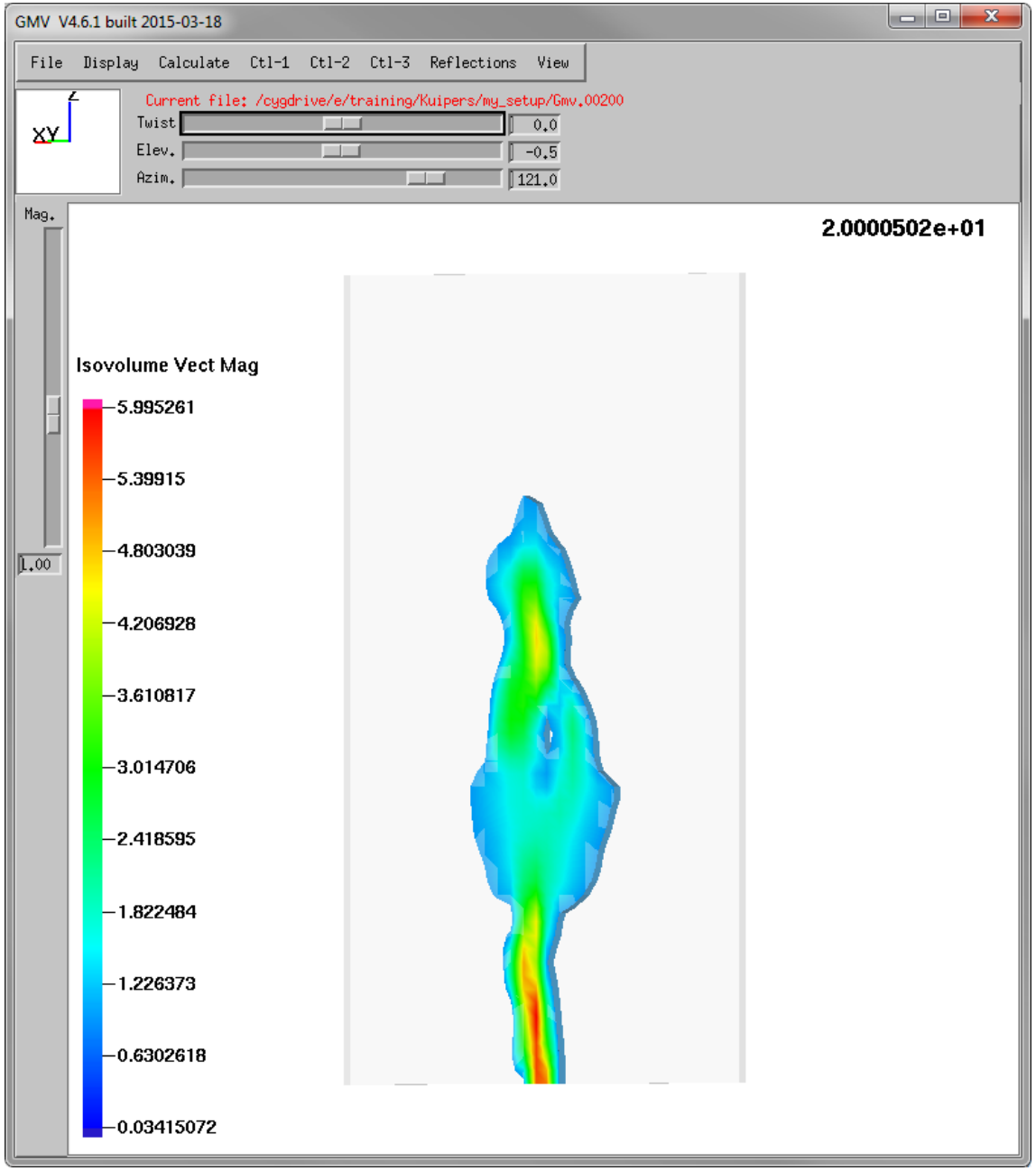
Isovolumes

- It is useful to change the field of the isovolume to identify regions with specific criteria
  - Where does the gas velocity exceed 5x Umf?
  - Where is the mass fraction of a species greater than some tolerance?
  - Where are the voids in the bed?
  - Where are the particles moving up? Where are they moving down?
- Click on **Calculate →Isovolume**
- In Isovolume window, click on **New Field**
- Select **Vect Mag** as new field
- Set the **Isovolume Range** from 1-10



Isovolumes

- With the range from 1-10, areas of where gas velocity is high are shown.
- The range can be changed (4-10 or 5-10) in order to show the areas with peak gas velocity

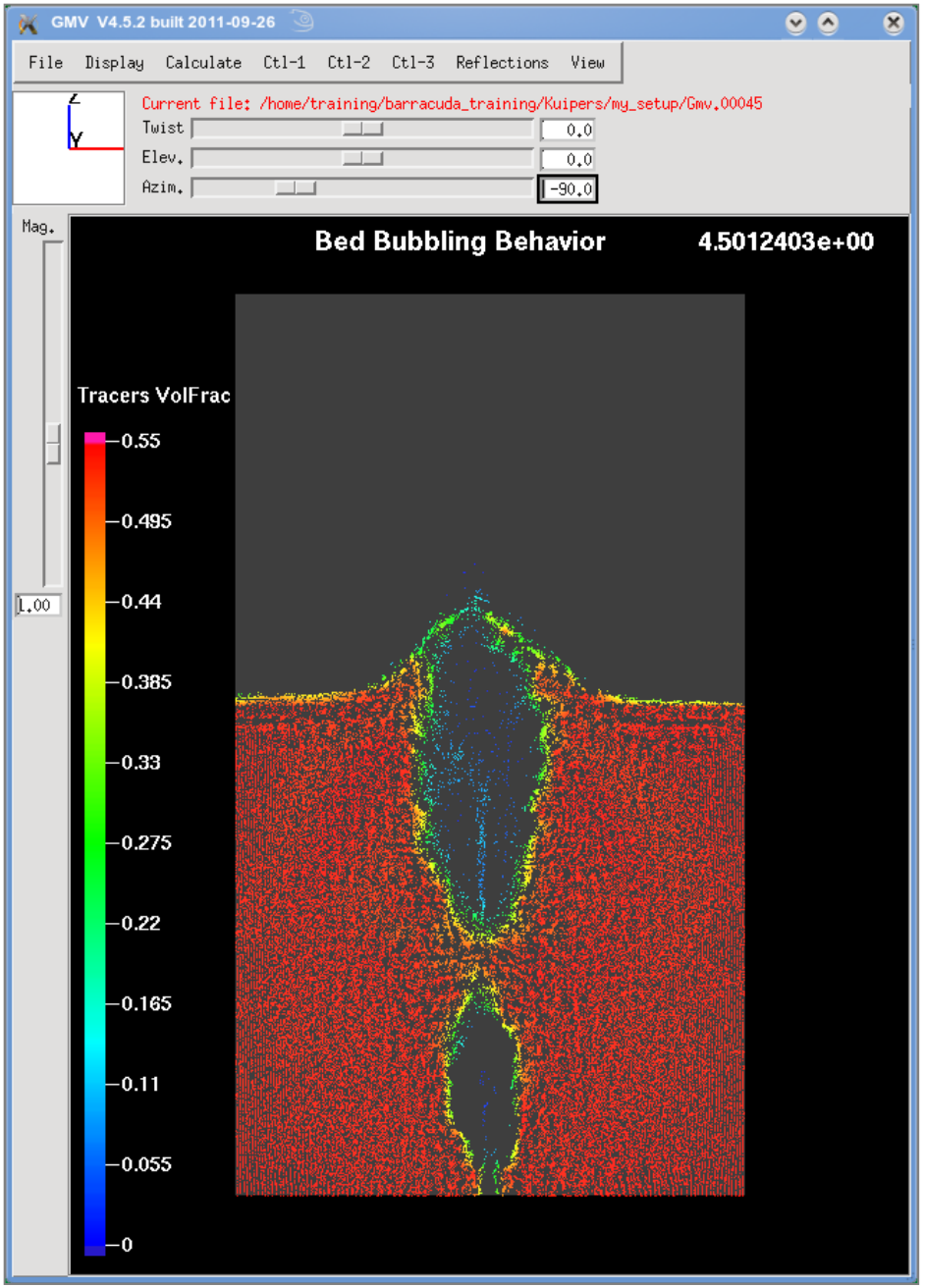


Part III: Post-Processing Assignments

- Take a moment to review the post-processing material covered up to this point
- Ask your instructor if you have any questions
- On your own, complete the following three assignments

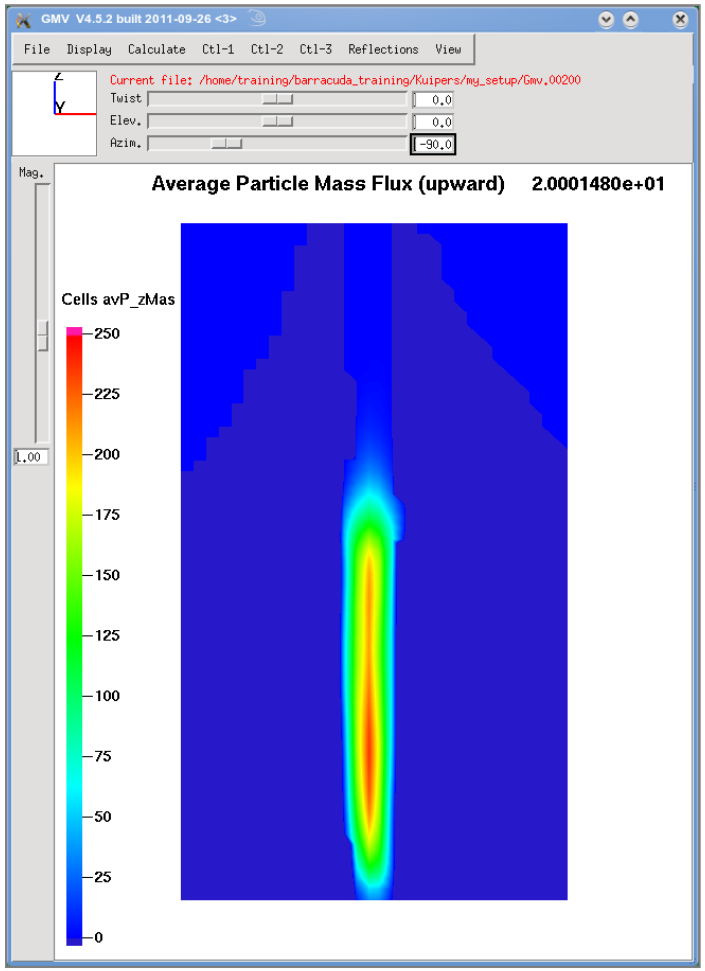
Post-Processing Assignment: Transient Bed Behavior

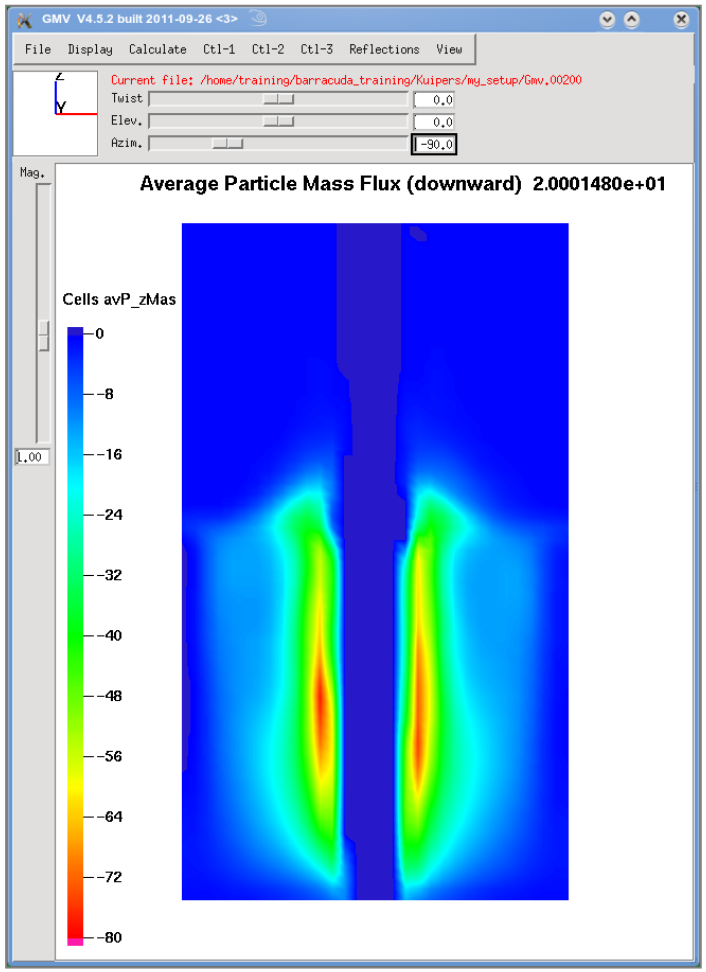
- Create an animation of the bed bubbling behavior
- Use BATCHMOVIE.sh as covered earlier in this presentation.



Post-Processing Assignment: Average Particle Mass Flux

- Create these images of the average particle mass flux in the z-direction (avP_zMas)
- How do they provide additional information on particle movement in the bed?





Post-Processing Assignment: Average Particle Mass Flux Vectors

- Create these images of the time-average particle mass flux vectors in the positive and the negative z-direction

- **Hint**: Build your own vectors and use the Select feature in the Nodes window to view the vectors in the positive z-direction separately from the vectors in the negative z-direction