Kuipers Training Problem Part 2: Project

August 2020

CPFD Software
1255 Enclave Parkway, Suite E
Houston, TX 77077 USA
+1 (713) 429-1252
www.cpfd-software.com

Setup

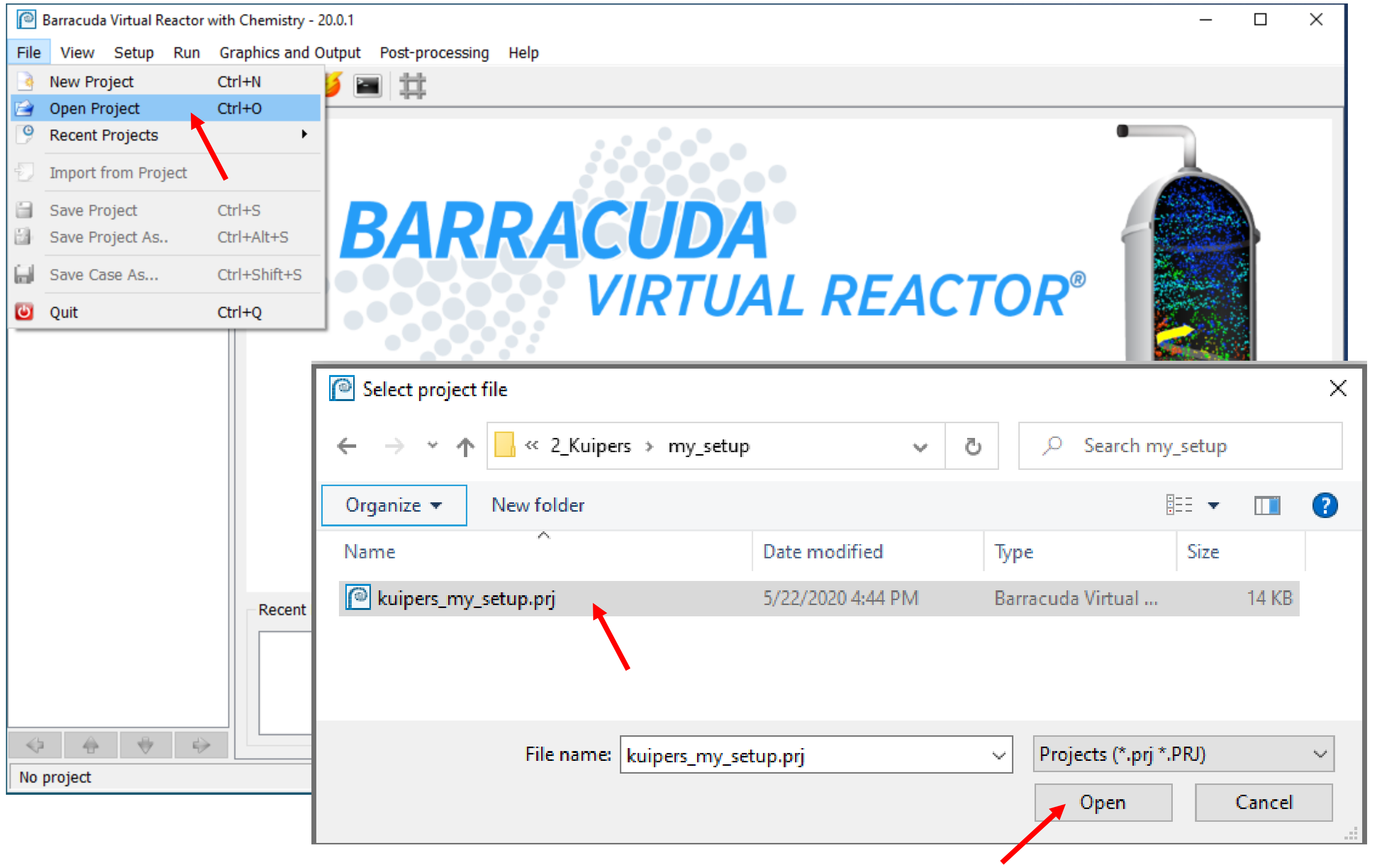
Opening a Project File

Launch Barracuda:

- Double-click on the desktop icon
- Click on File → Open Project
- Navigate to the training directory, and go into the folder: `2_Kuipers/my_setup/`
- Choose the project file: `kuipers_my_setup.prj`
- Click Open

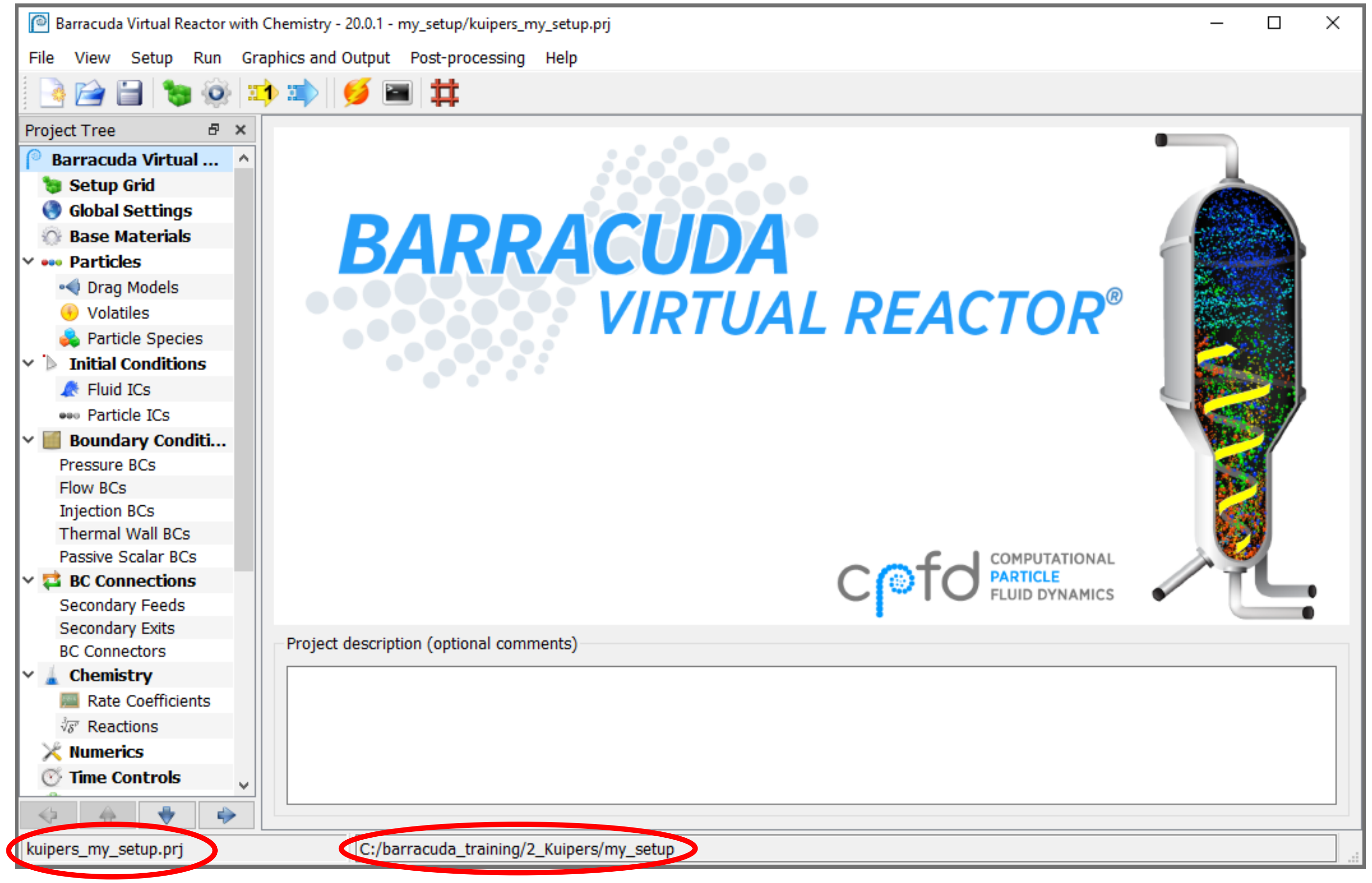
Suggested location for the Barracuda training directory:

- Linux: `~/barracuda_training/`
- Windows: `C:\barracuda_training\`



Navigating the Barracuda GUI

Notice the project file and working directory are listed at the bottom of the main Barracuda GUI window



Setup Grid

Barracuda simulates fluid-particle behavior by dividing the physical domain into a 3D computational grid.

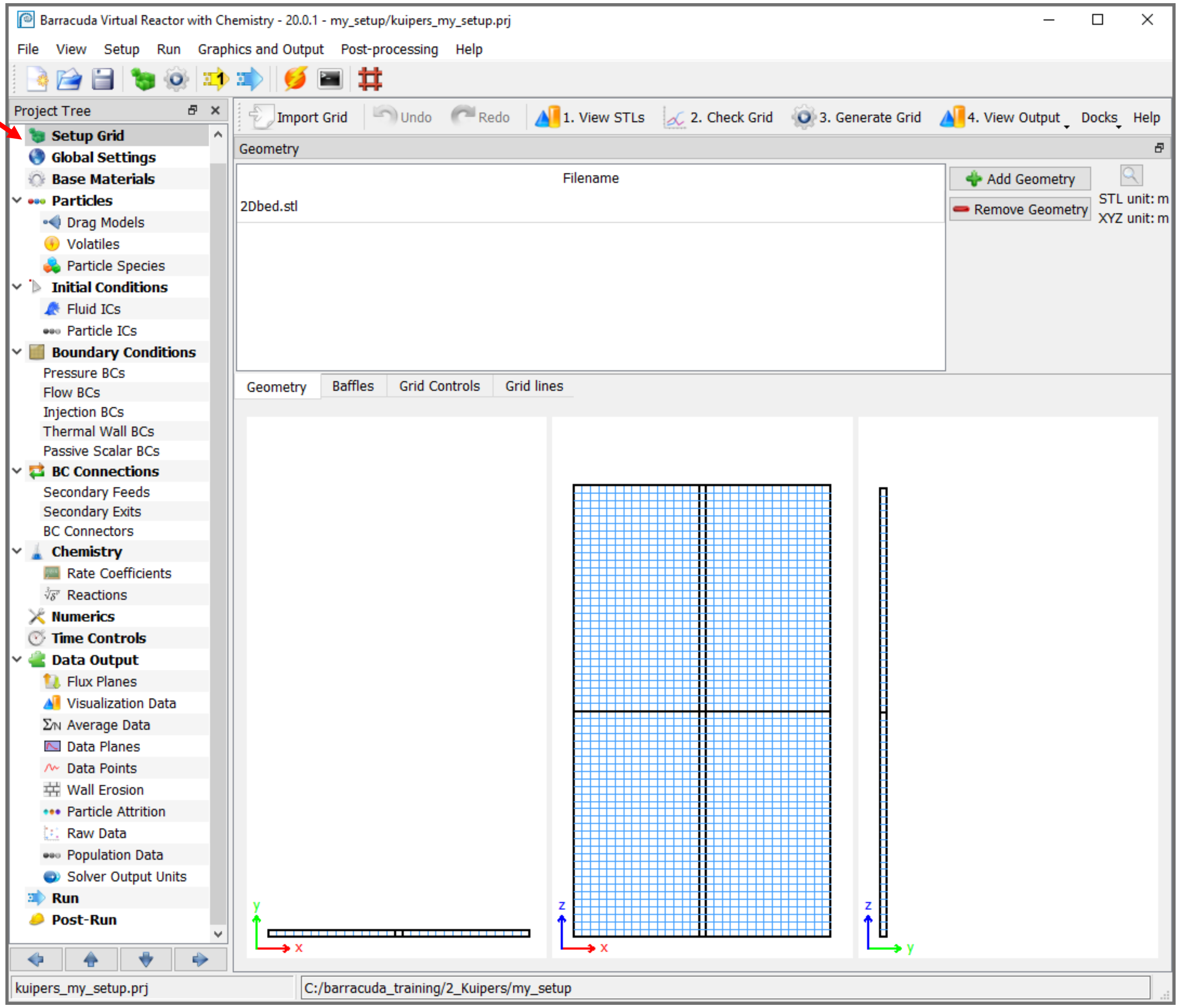
Each cell within the grid provides a location for the solver to calculate Eulerian values:

- Pressure
- Temperature
- Velocity
- Composition, etc.

The grid also provides a framework for specifying boundary conditions within a simulation.

Adding cells to a simulation will increase the resolution and often accuracy of the solution, but also increase the computational time required.

For the Kuipers problem, gridline locations are already provided in the `kuipers_my_setup.prj` file.

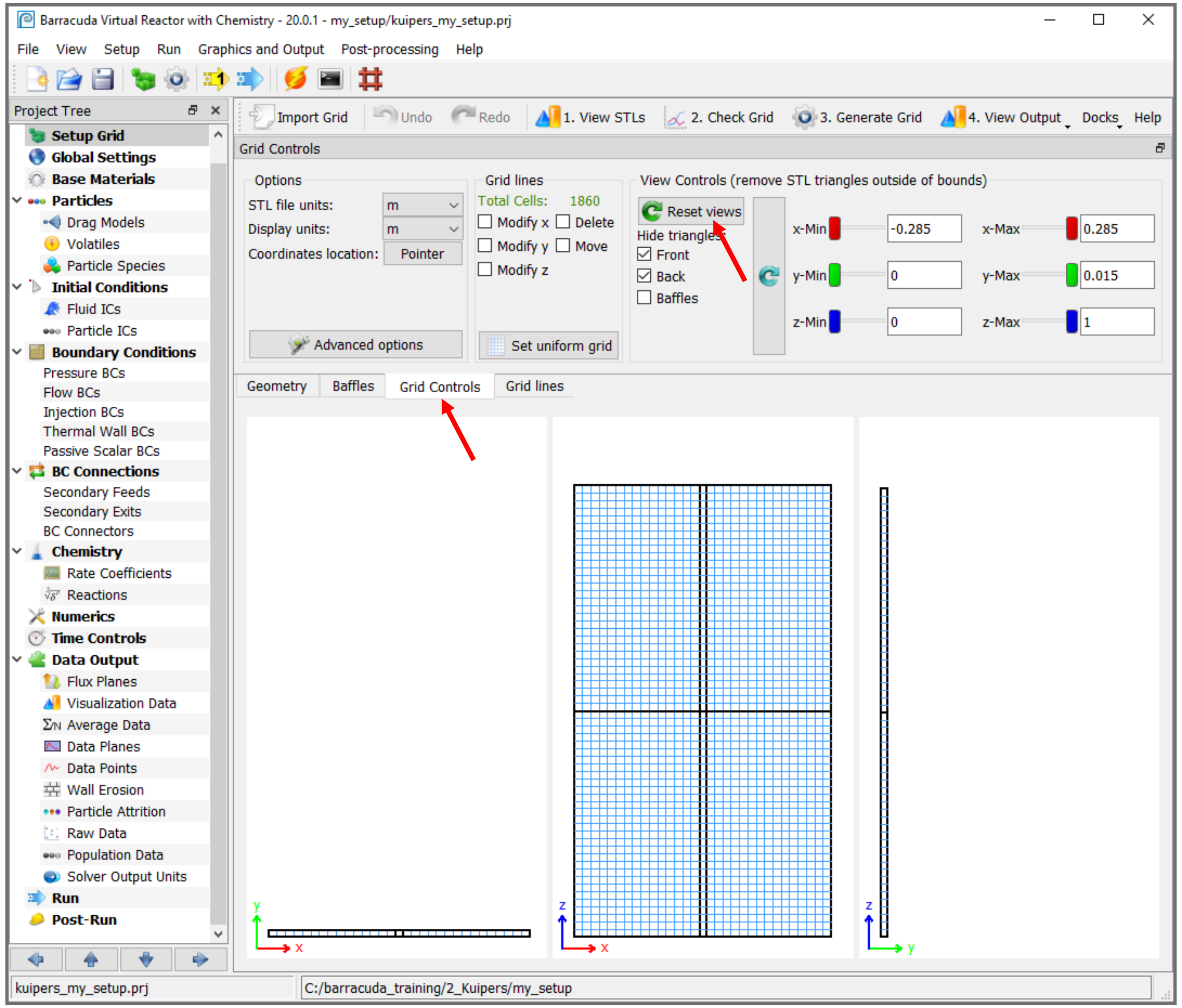


Grid Controls

Using the mouse:

- The grid may be translated using the center mouse button
- Zoom in/out is accomplished holding the right mouse button and moving it up/down
- Alternately, the scroll wheel can also be used for zoom
- Note: each of the panel views can be panned and zoomed independently

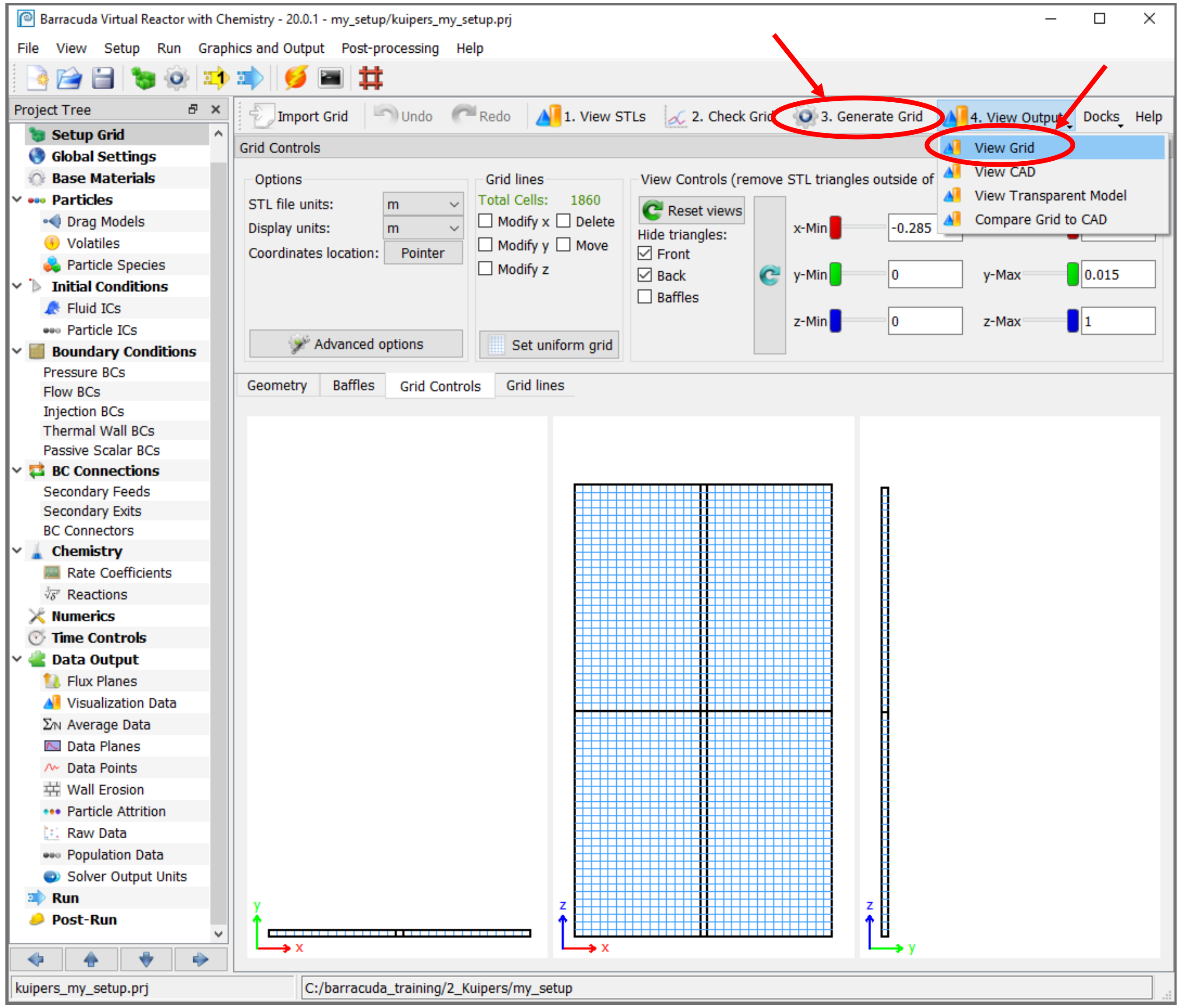
Reset views button in Grid Controls tab will return pane views to default position



Generating and Viewing the Grid

The computational grid will be generated based on the gridlines and STL file

- Click Generate Grid
  - This automatically saves the project file
- Once the grid generator runs, open the grid using the View Output button and selecting View Grid

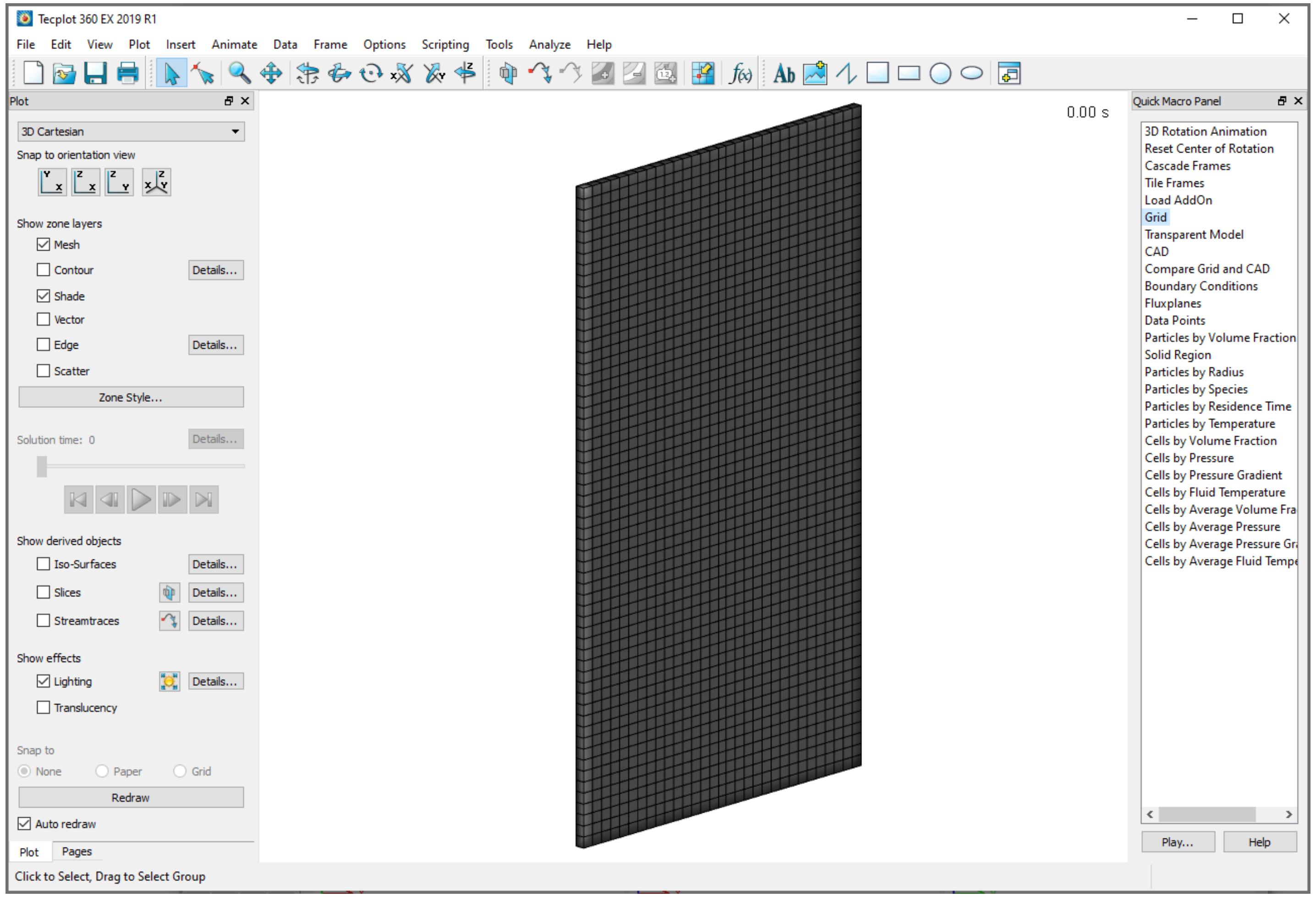


Viewing the Grid

The grid can be viewed using Tecplot

To rotate, translate, or adjust the zoom on the grid, use the buttons in the top tool bar.

- Watch Tecplot's video [Tecplot 360 Basics: Rotate and Zoom]

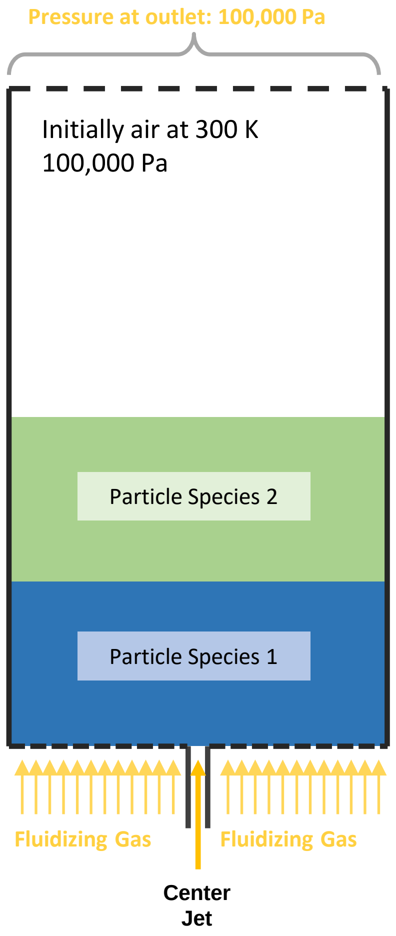


Note:

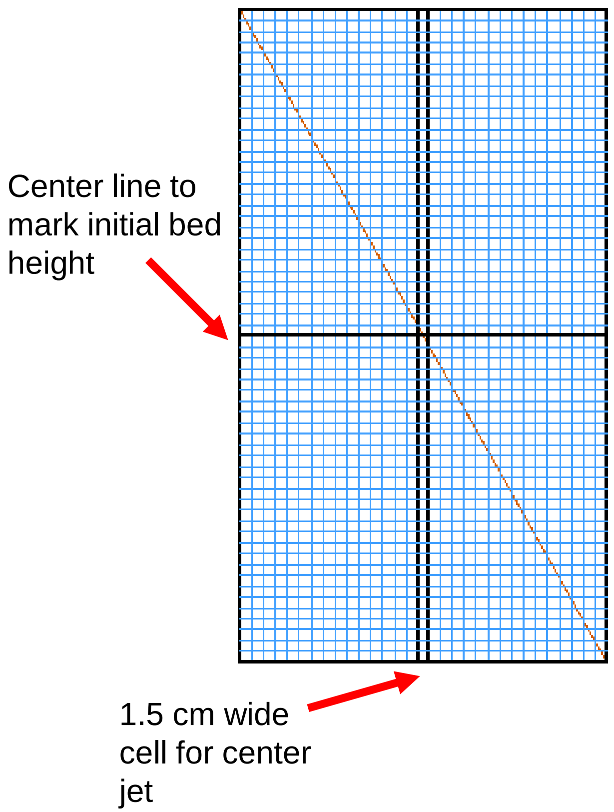
- This is the coarsest grid possible for this problem (resolving the center jet with a single cell).
- This is not necessarily the recommended grid resolution, but rather intended to illustrate how the CPFD method obtains resolution from both the computational cells (grid) and computational particles (gridless).

Overview of the Kuipers Bed Grid

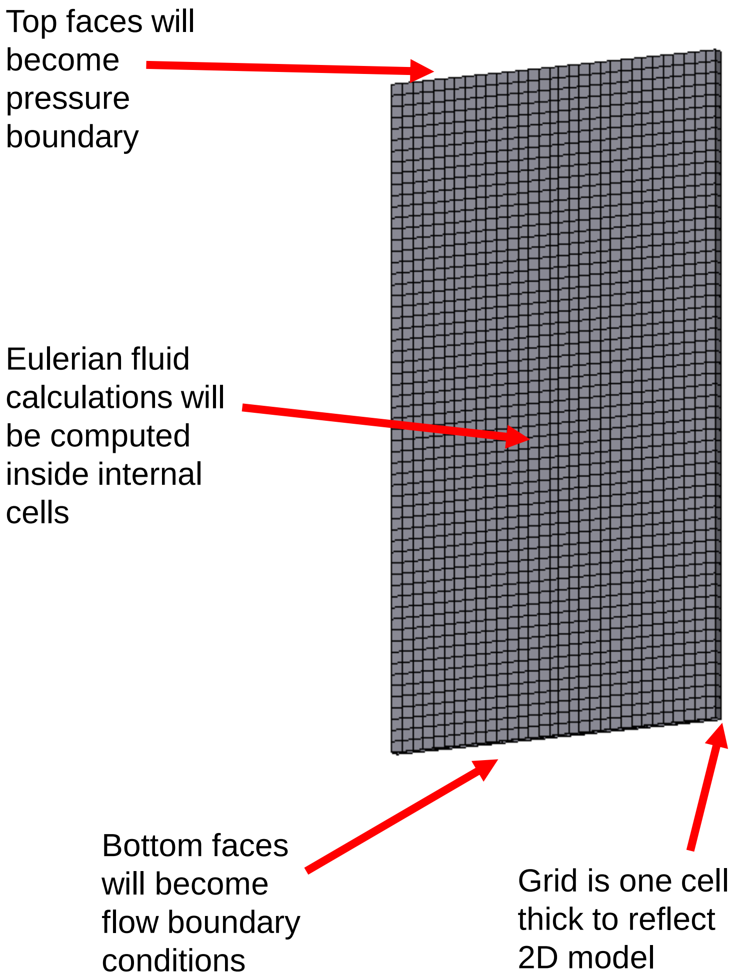
Physical Setup



Grid Setup



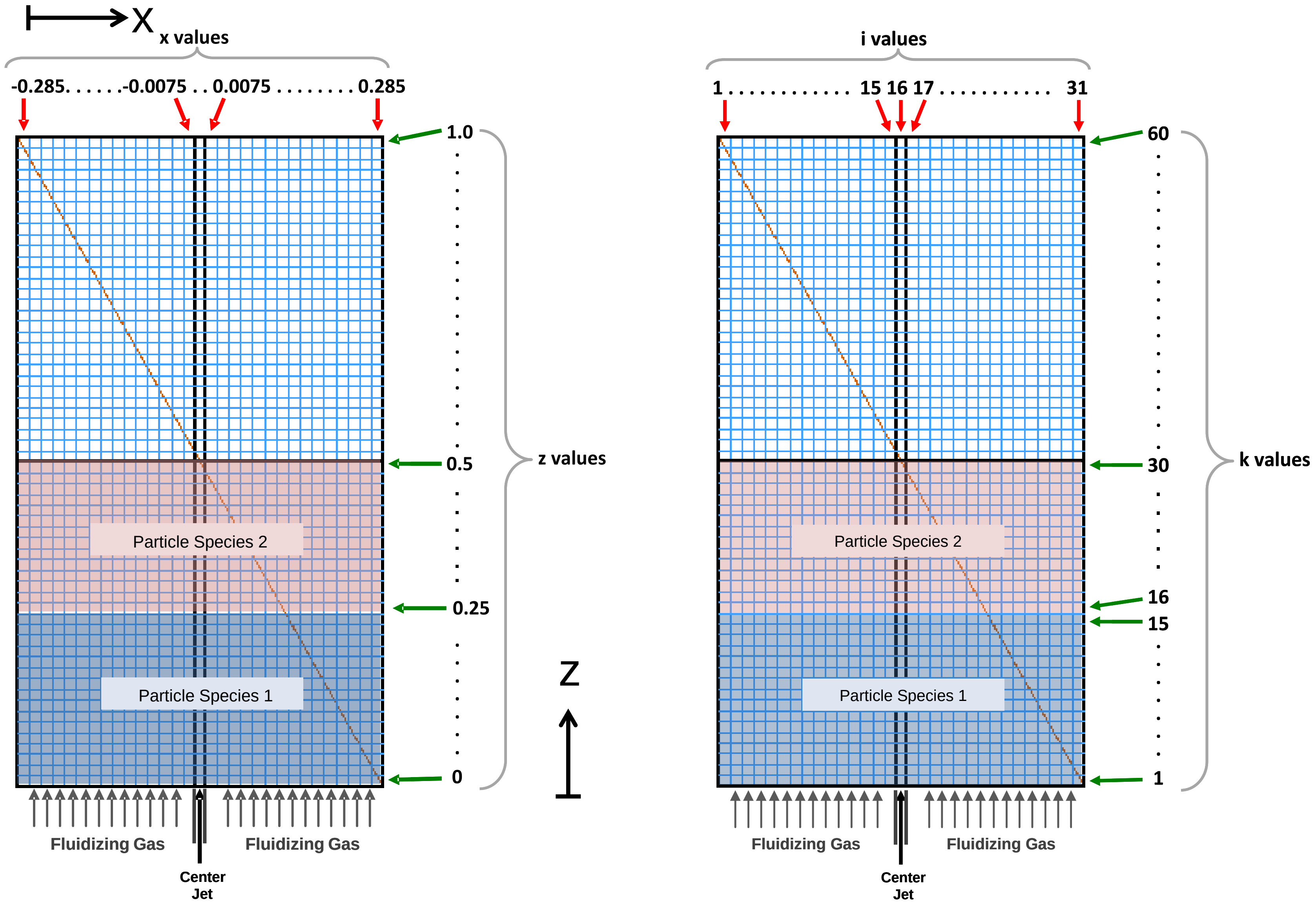
Generated Grid



Grid Coordinates

Grid cells are located by x,y,z values

Cells indices are an alternative way to reference locations in a Barracuda model. Each computational cell is identified by a unique i-j-k coordinate, which reference the cell in the x-, y-, and z- directions, respectively.



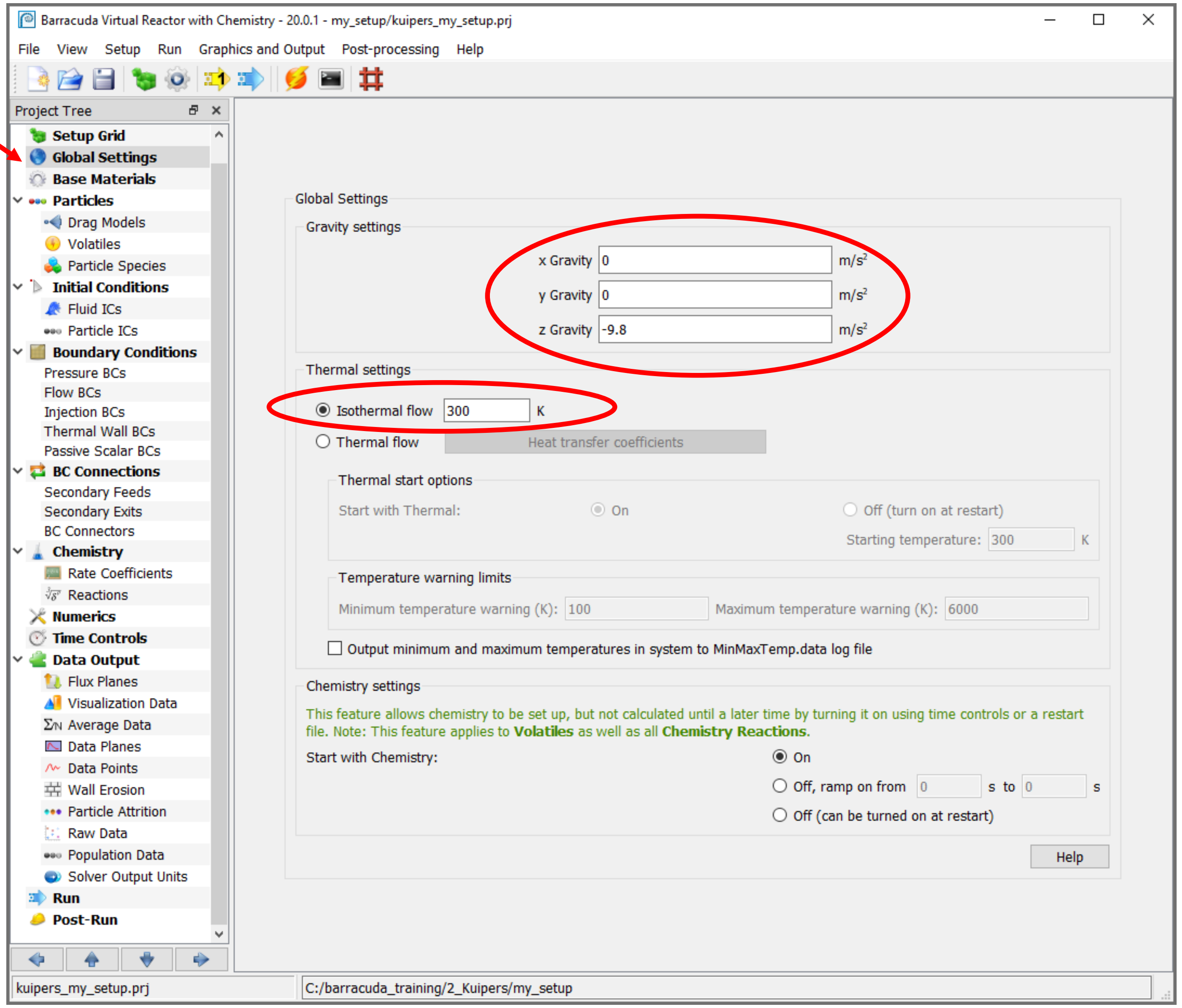
Global Settings

Click on Global Settings

Set Gravity vector

- In this example, gravity is in the negative z direction
- Notice the vector magnitude is 9.8 m/s^2

Isothermal flow should be selected.



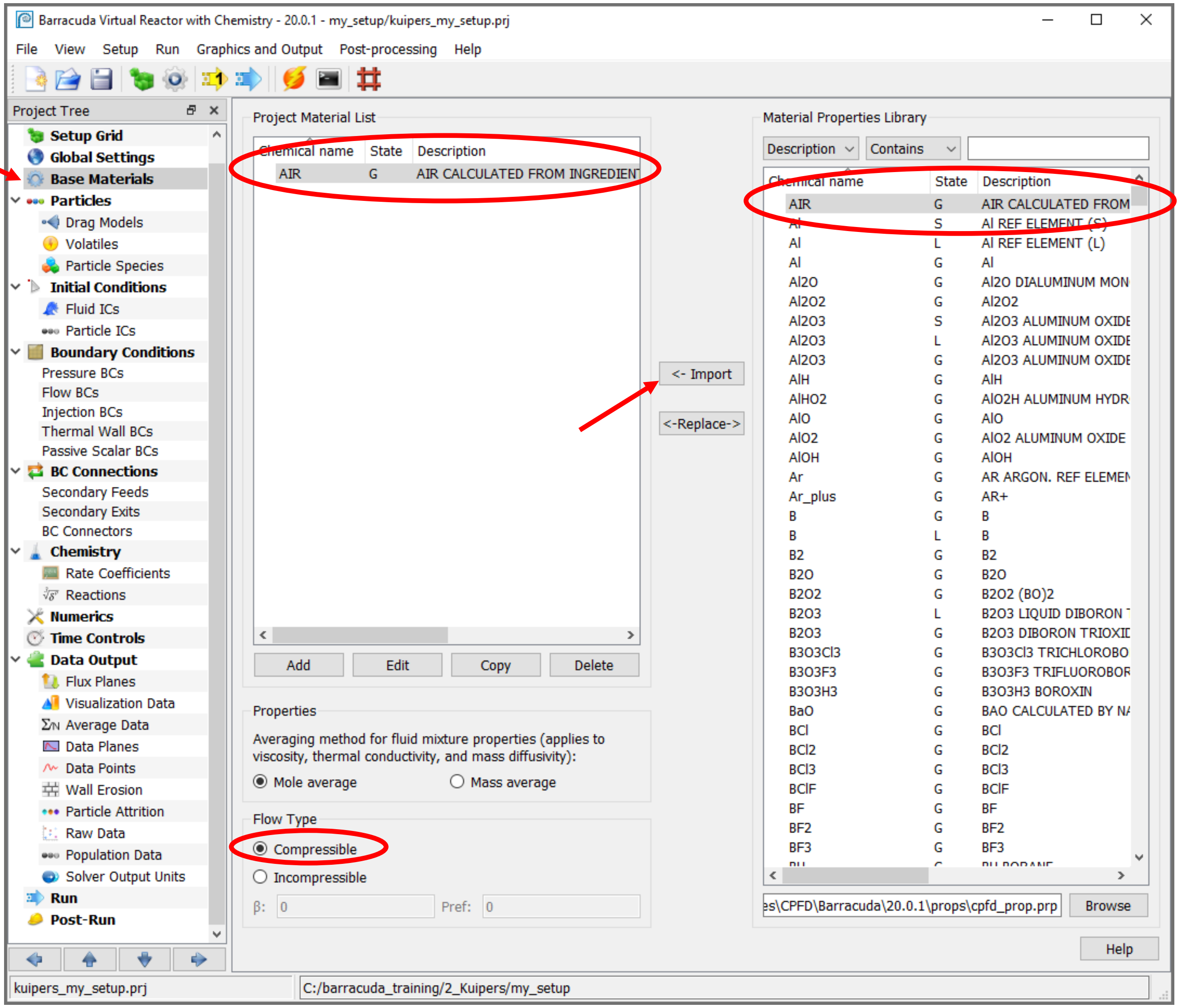
Base Materials

Click on Base Materials

- All project materials, fluids and solids, must be defined here
- Project materials can be imported from the Material library OR new materials can be created with the Add button

To add the fluid:

- Select AIR from the Material Library
- Click Import
- AIR now appears in the Project Material List at the left
- Verify that the Flow Type is set to Compressible since the fluid is a gas



Base Materials

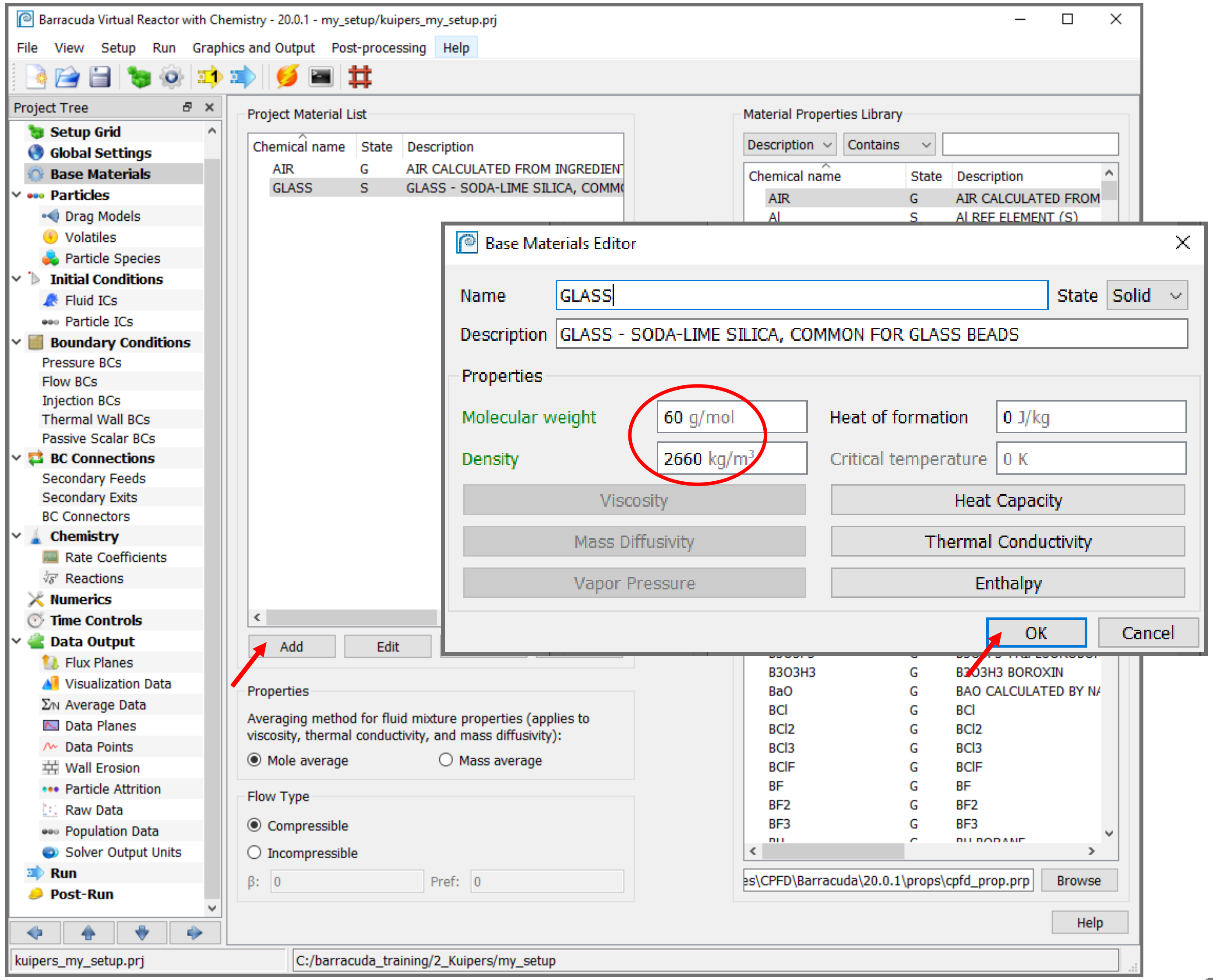
All material properties can be defined and edited in the Base Materials window. To define the solid material:

- Click Add
- In the Material Properties window, enter the Chemical name as GLASS.
- Enter an appropriate description
- Set the State to Solid
- Set the Molecular weight to 60 g/mol
- Set the Density to 2660 kg/m3
- Click OK

GLASS now appears in the Project Material List at the left

NOTE:

- Red writing means that data needs to be input, green writing means that the property is ready, black writing means that the property is not needed for the calculation.
- Other material property data such as thermal conductivity, heat capacity, and heat of formation would have to be specified if this were a thermal or reacting problem. Viscosity is required if the new material is a fluid.

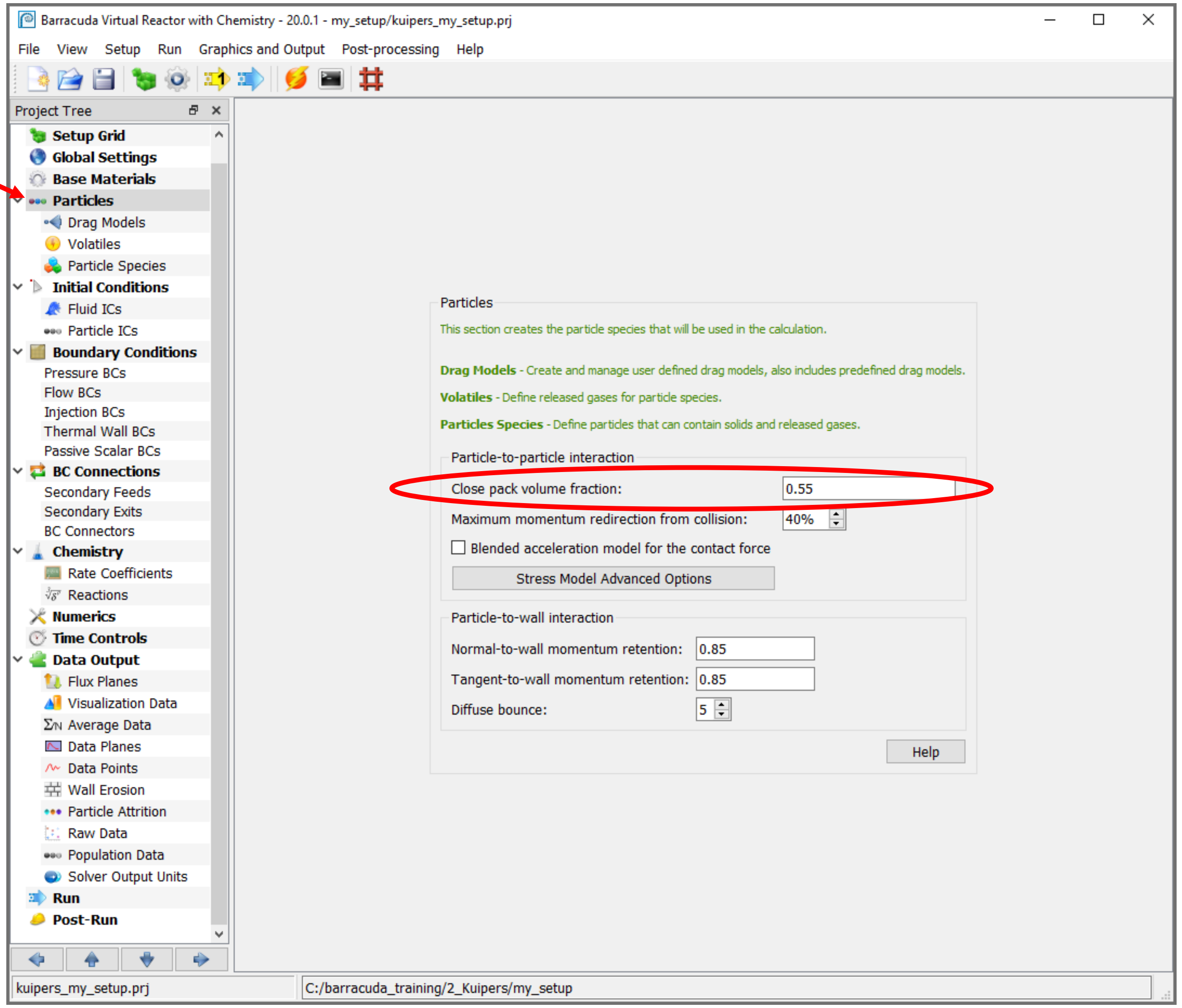


Particles

Click on Particles

Enter a Close pack volume fraction of 0.55

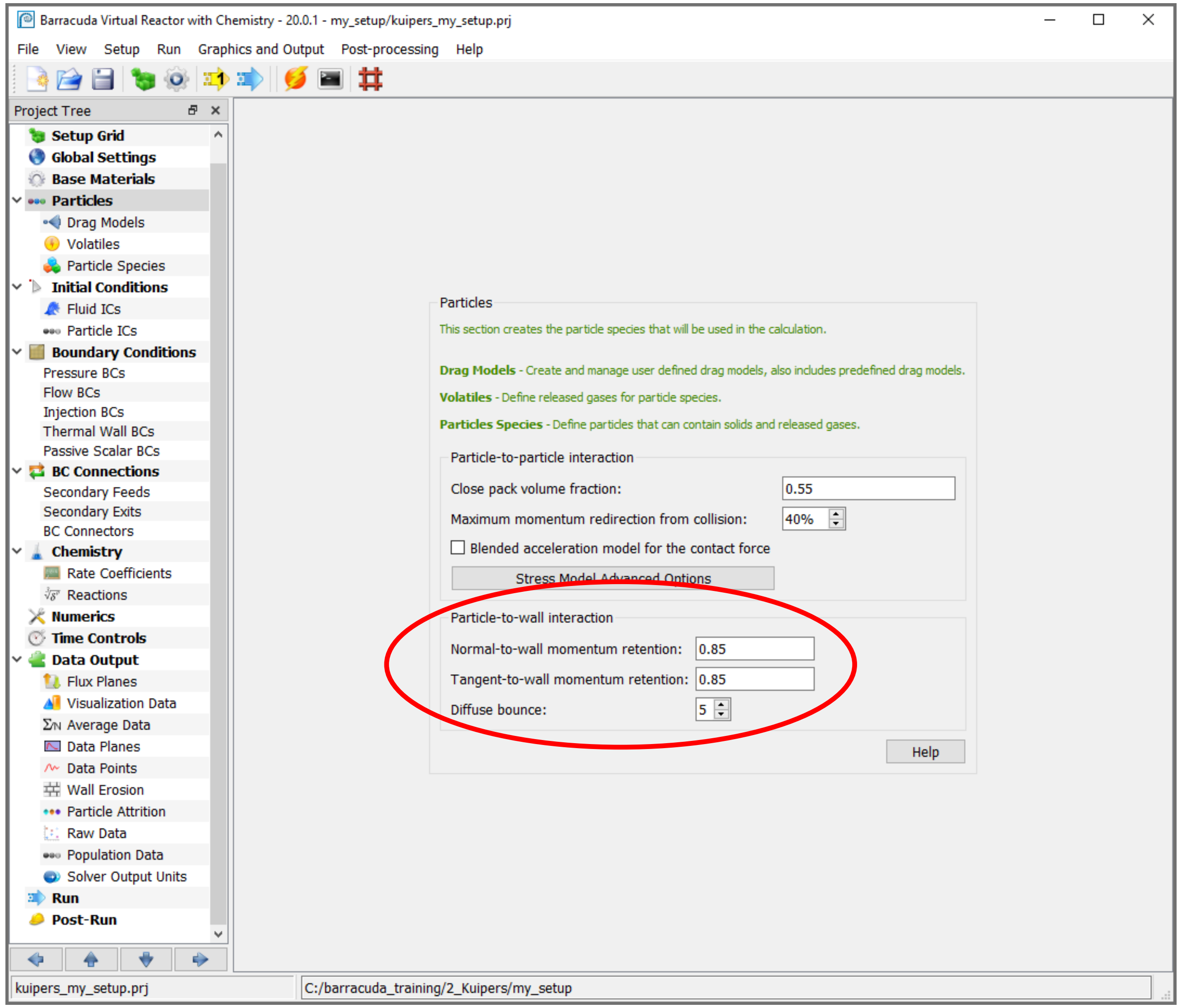
- This is the maximum amount of solids packing permitted in a cell
- This means that up to 55% of a reasonably-sized control volume can be occupied by particles. Conversely, at least 45% of the same volume must be occupied by the fluid.



Particles

Set Particle-to-wall interaction properties:

- A Normal-to-wall momentum retention coefficient is the maximum normal component of particle momentum which can be retained after the particle “bounces” off a wall.
  - Set this to 0.85
- A Tangent-to-wall momentum retention coefficient is the maximum tangential component of particle momentum which can be retained after the particle “bounces” off a wall.
  - Set this to 0.85
- A Diffuse bounce coefficient is the amount of particle scatter after the particle “bounces” off a wall
  - Set this to 5, which is the maximum value

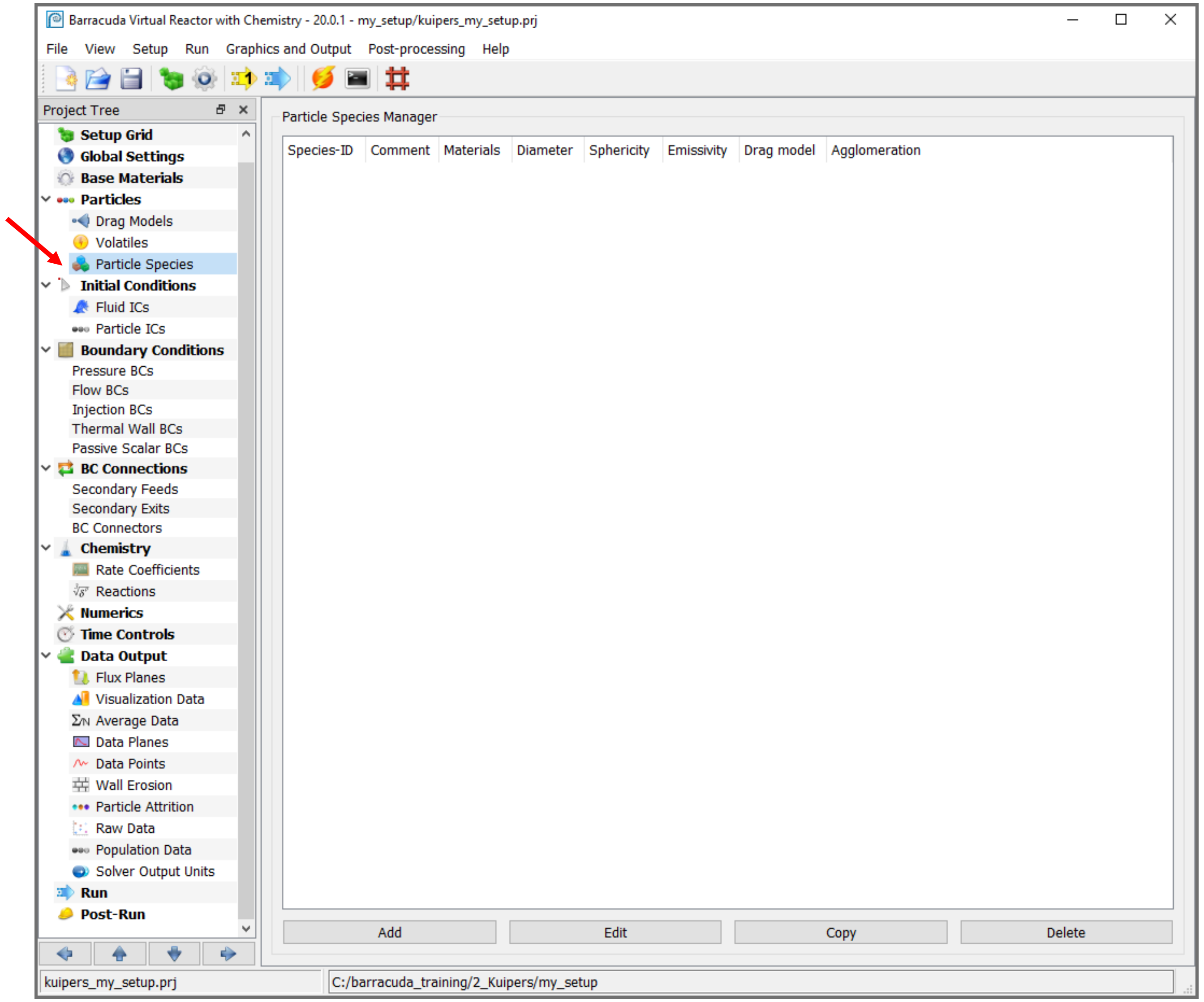


Particle Species

Click on Particle Species

We have only one type of solid particle (GLASS)

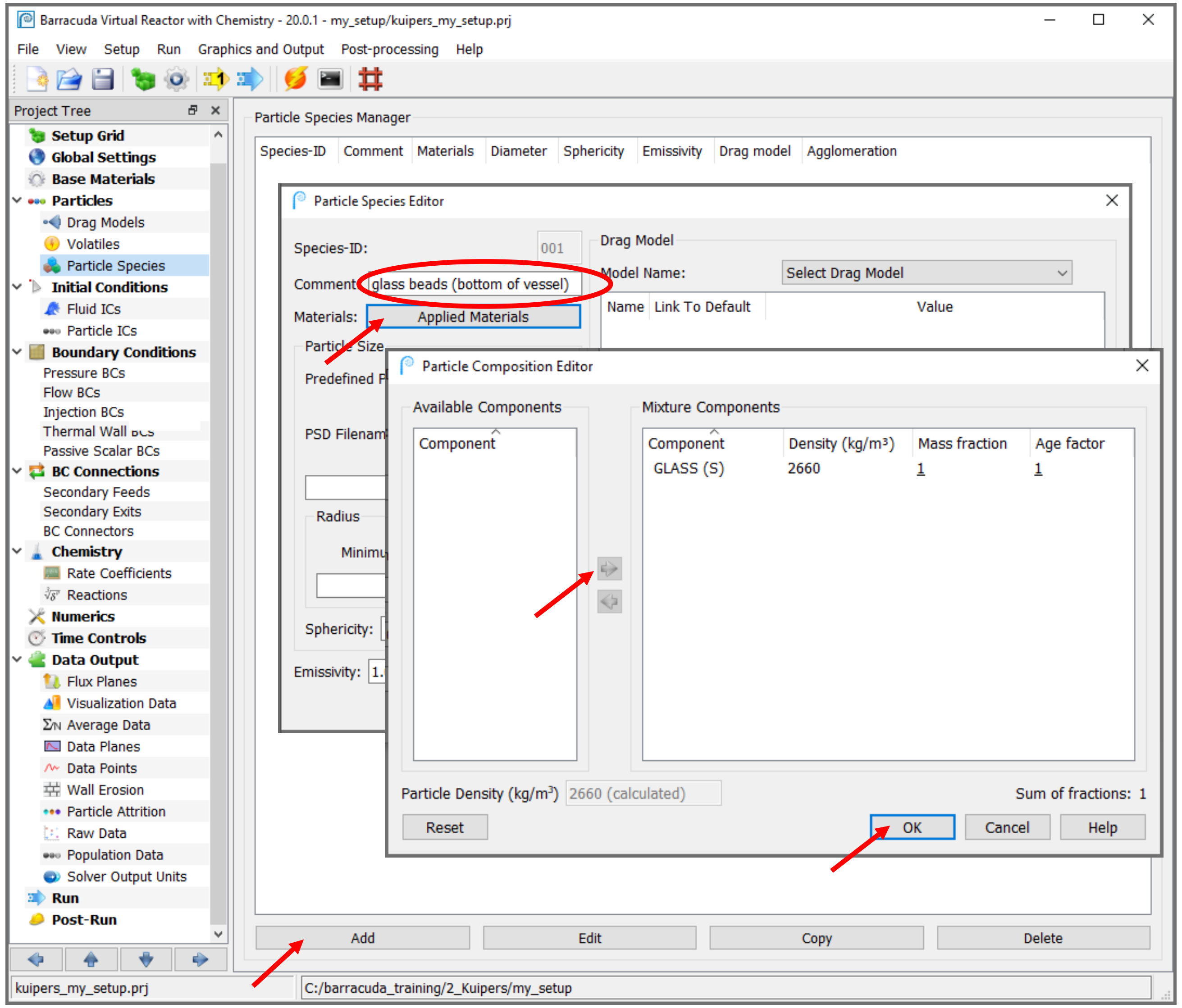
- However, we want to use two colors (one in bottom half of bed, another in top half) to view mixing
- To accomplish this, we must define two separate species of the same material (GLASS)



Particle Species – Applied Materials

To add the first particle species:

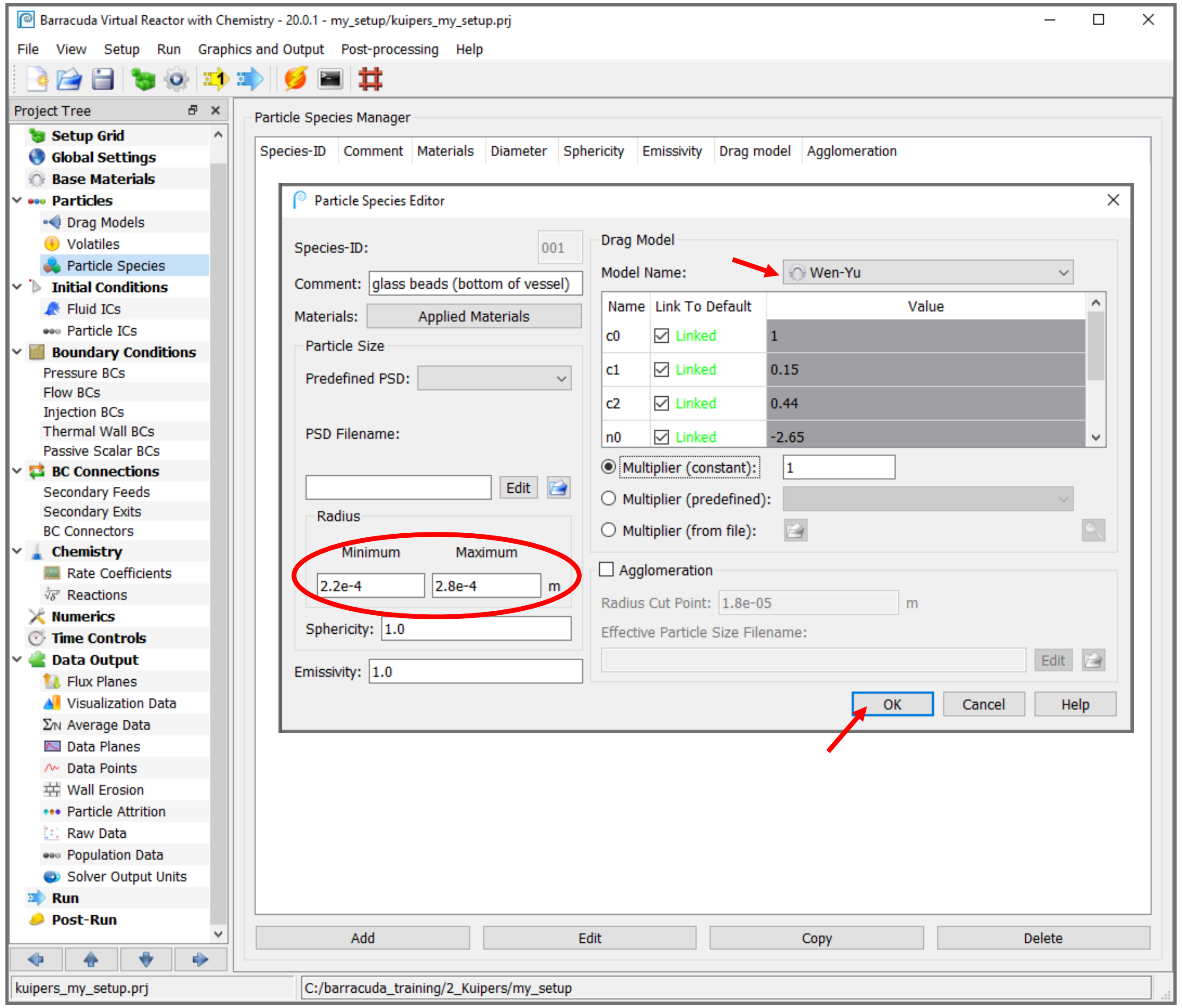
- Click on Add
- Add a description in the Comment section of Particle Species Editor
- Click on Applied Materials
- In the Particle Composition Editor import GLASS from the Available Components to the Mixture Components using the Right Arrow Button
- Click OK



Particle Species – Particle size and Drag Model

For the Kuipers setup, the particle diameters are between 440 and 560 µm

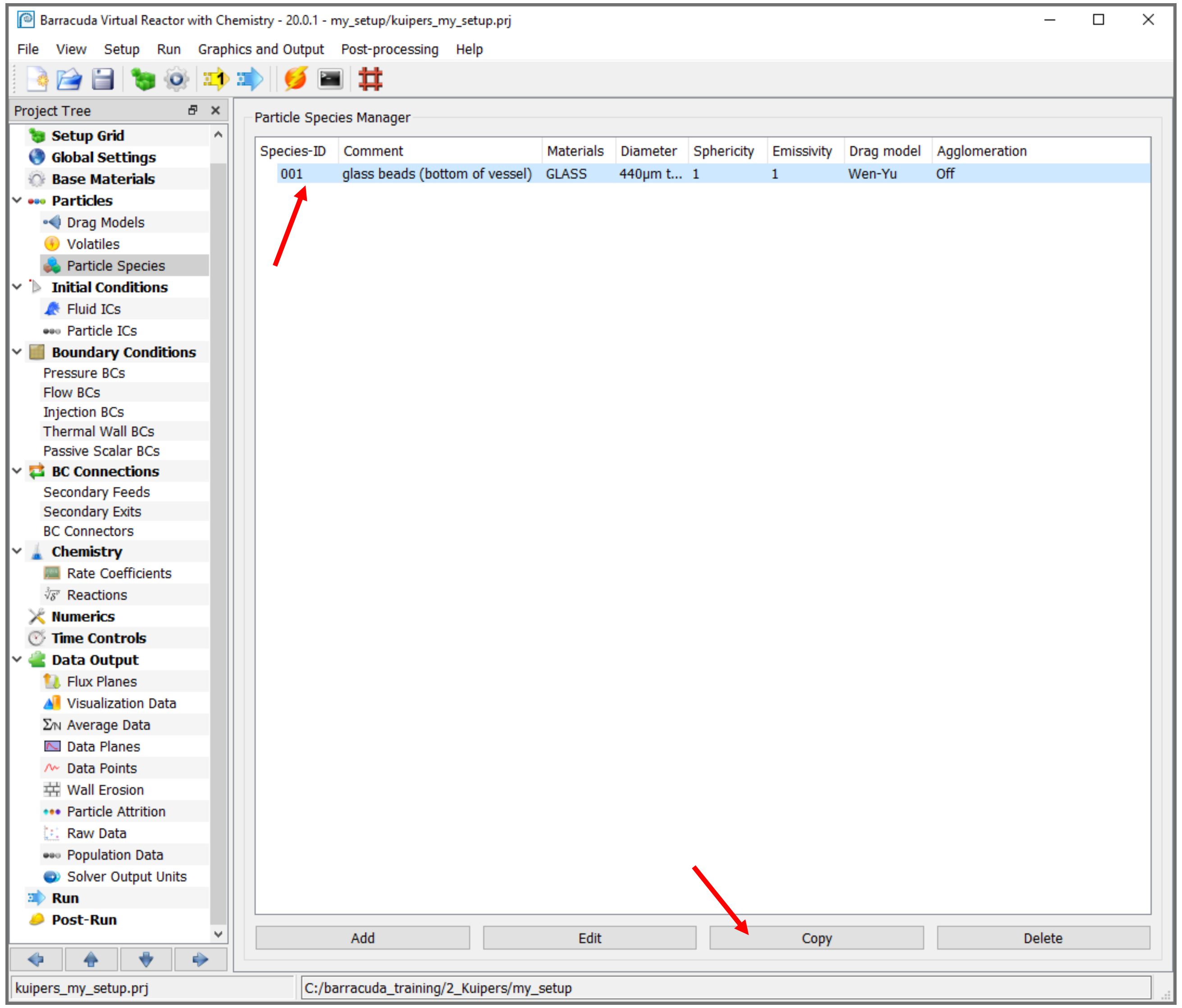
- Set the particle Radius from 2.2e-4 to 2.8e-4 meters
- Select Wen-Yu as Drag model in the drop down menu
- Click OK



Particle Species – Copy Species 001

To define a second, identical particle species:

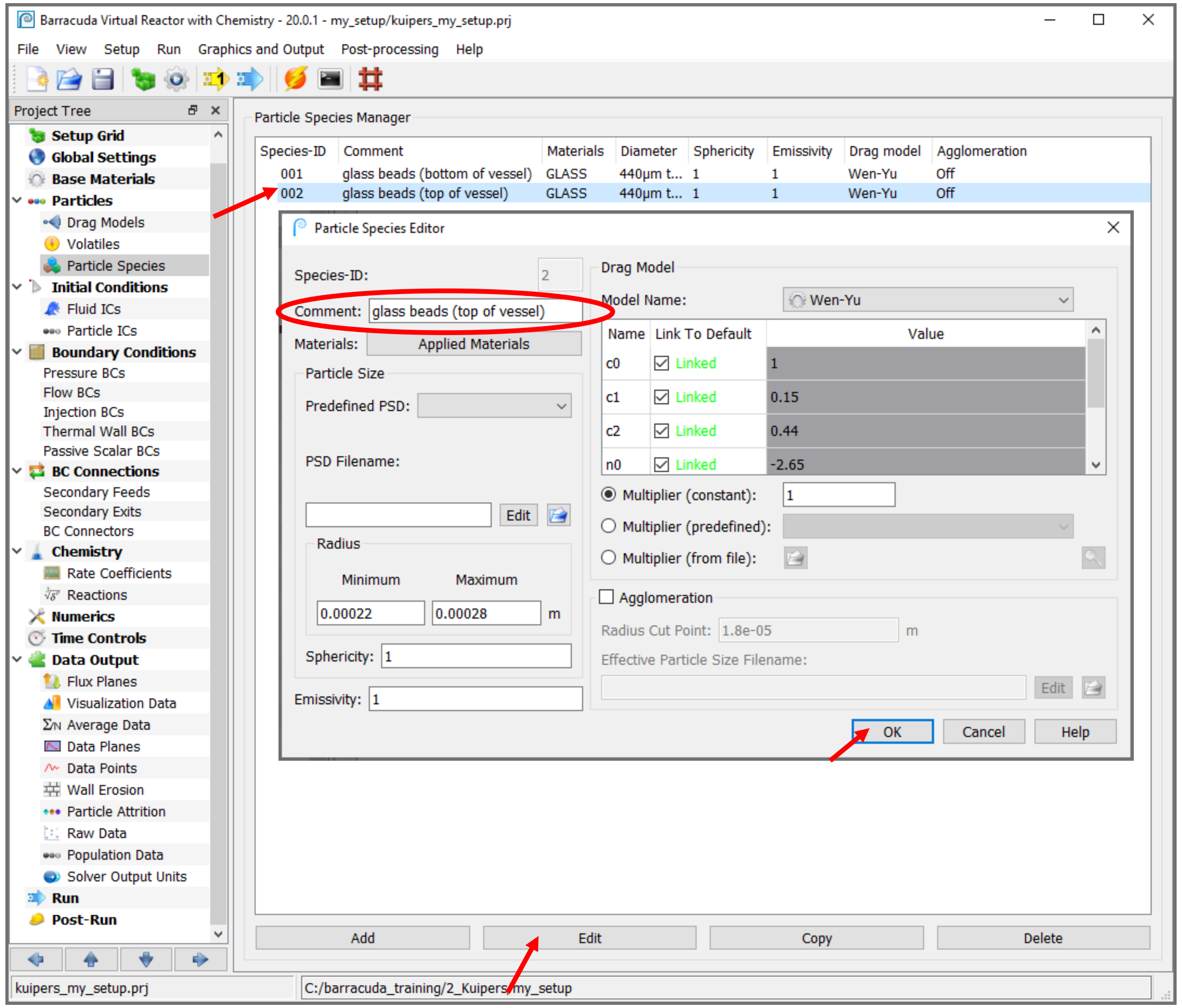
- Select species 001
- Click Copy



Particle Species – Edit Species 002

To change the comment describing the second particle:

- Select species 002
- Click Edit (or double-click on species 002)
- Change the Comment
- Click OK



Fluid Initial Conditions – Define Fluid

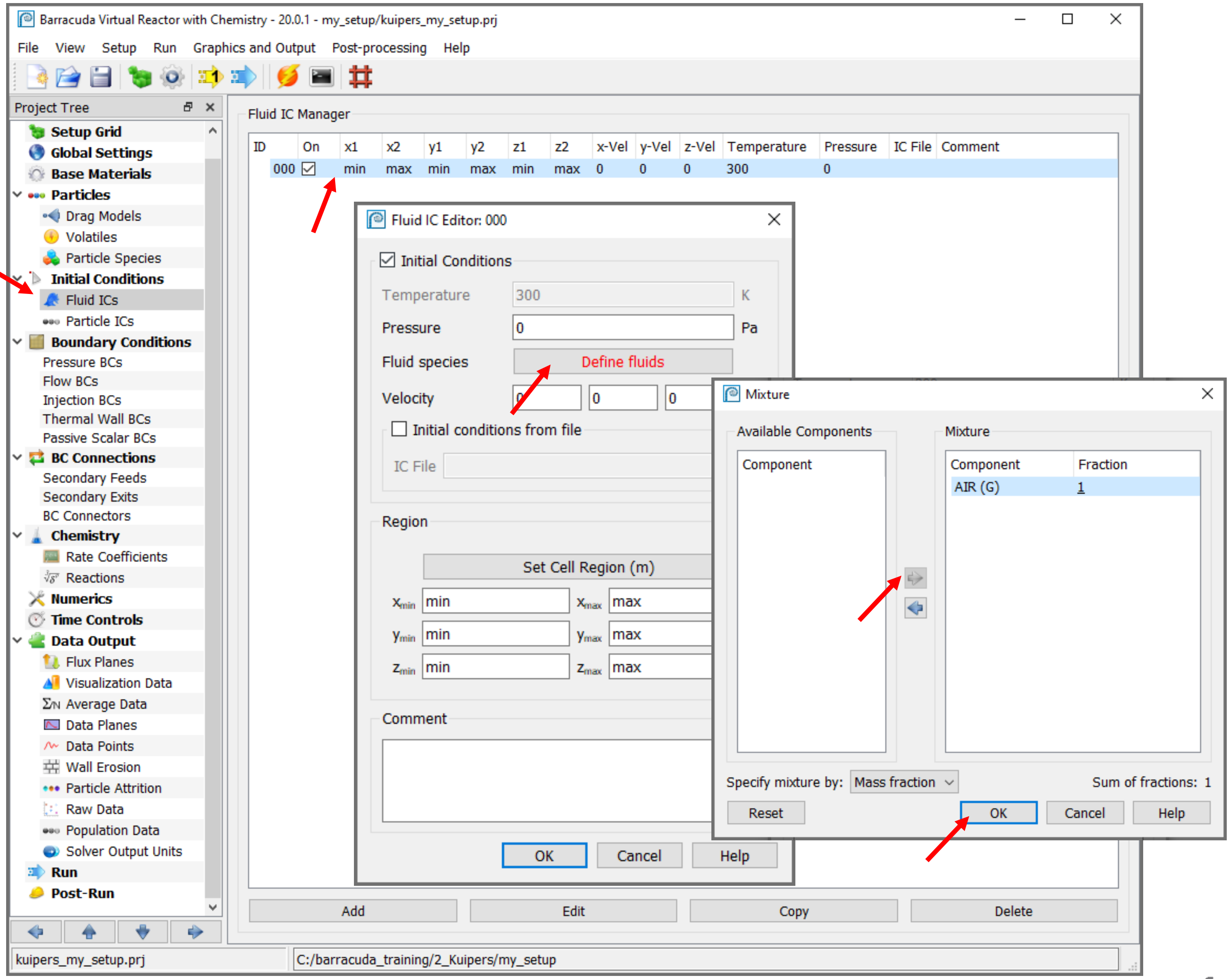
Click on Fluid ICs

Every Barracuda project has a default Fluid IC that needs to be edited:

- Double-click on the default fluid IC
- Click on Define fluids

In the Mixture window:

- Select AIR (G)
- Use the Right arrow button to import from Available Components to Mixture
- Click OK

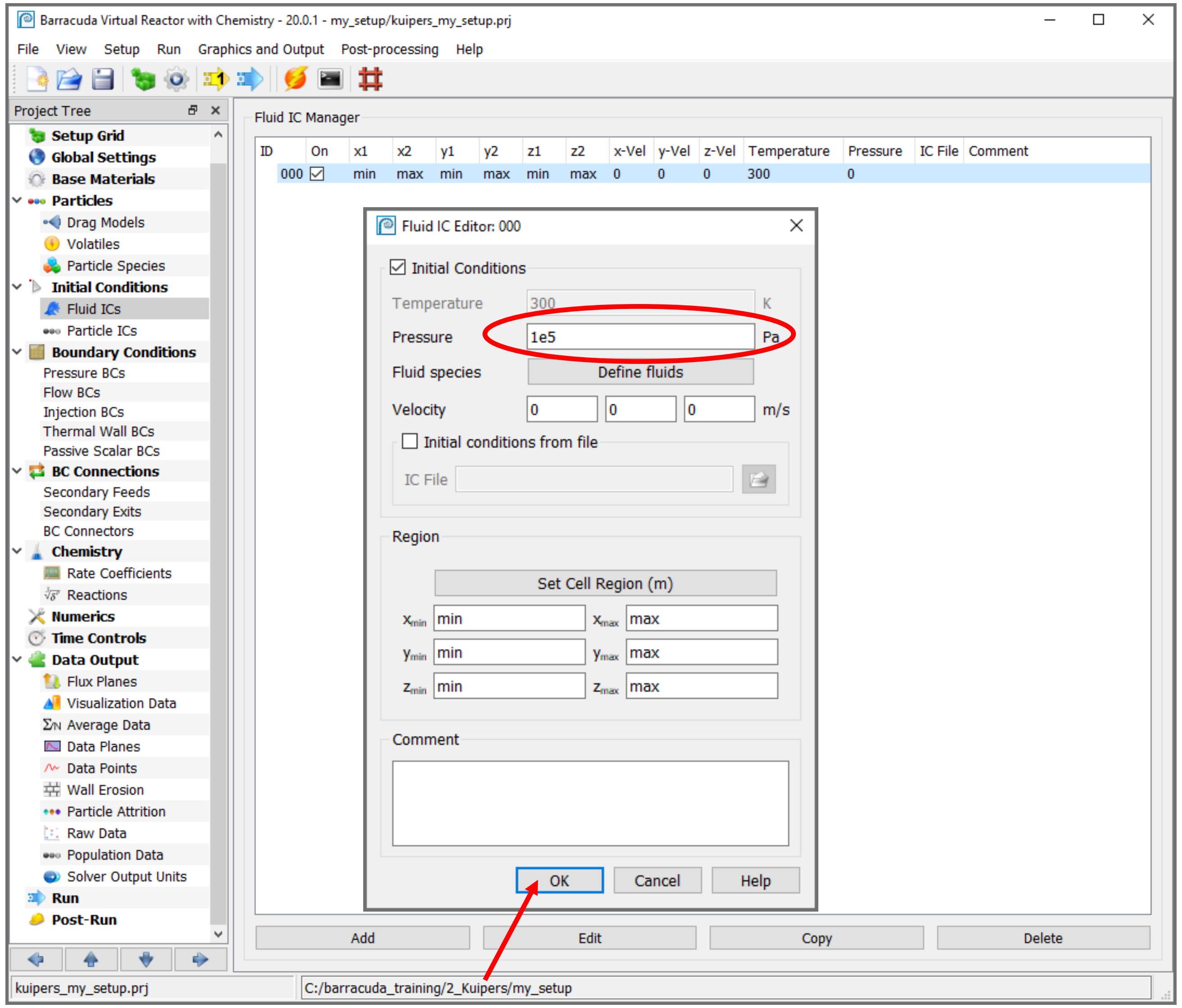


Fluid Initial Conditions – Set Pressure and location

For this example problem, air is initially at rest at atmospheric pressure.

In the Fluid IC Editor:

- Specify the pressure as 1e5 Pa
- Leave the x, y, and z fluid velocities at zero
- Leave the Region set from minimum to maximum for x, y, and z, since the air initially occupies the entire bed (area above and in between quartz particles)
- Click OK

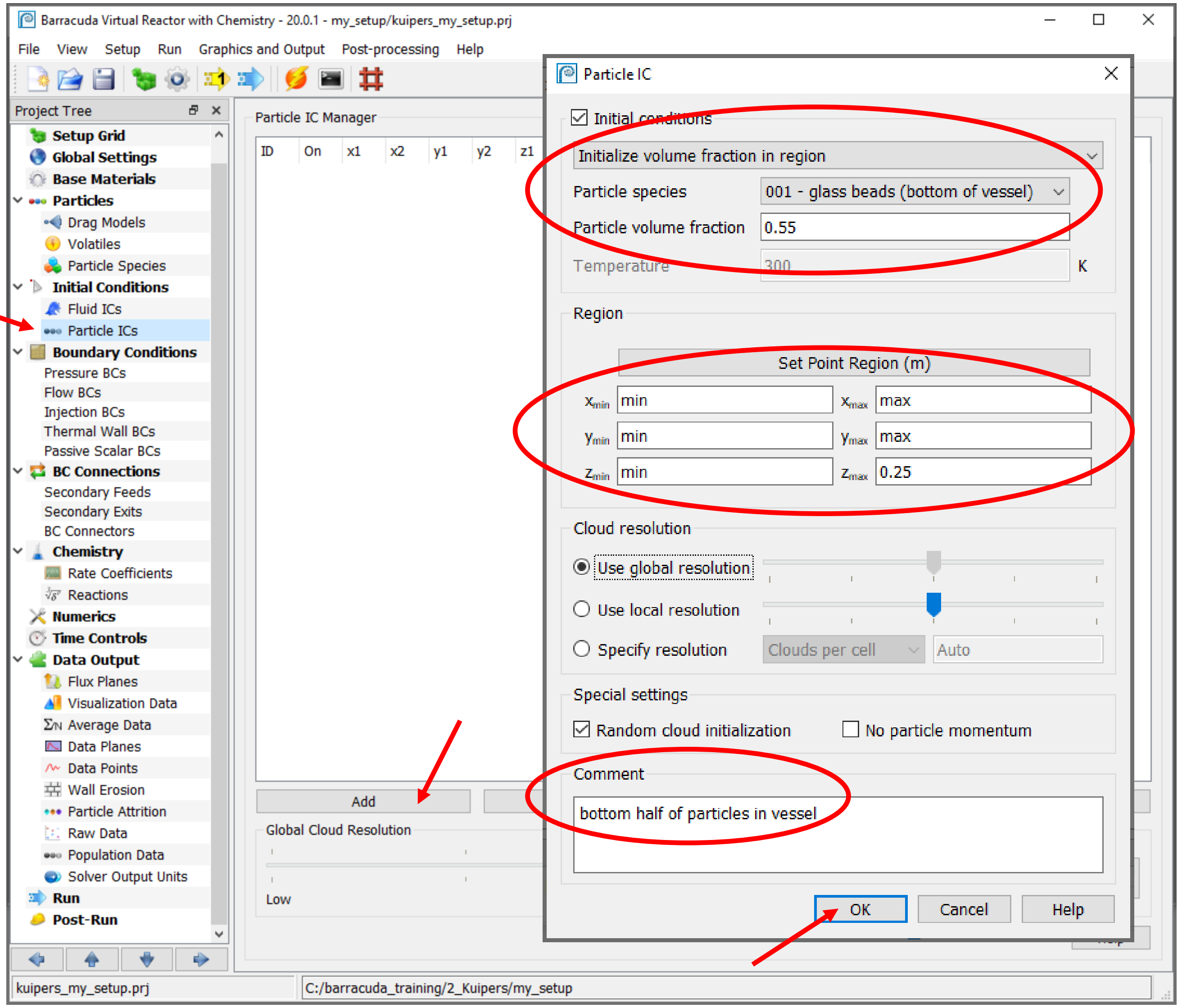


Particle Initial Conditions

To show mixing add two distinct layers of particles:

- Click on Particle ICs
- Click Add, which pops up a Particle IC dialog
- Select Initialize volume fraction in region
- Select 001 – glass beads in Particle Species
- Set Particle volume fraction to 0.55
- Set Point Region as shown in the table below. [= min,] = max
- Add descriptive comment
- Click OK

Boundary	x		y		z	
Species 1	[	]	[	]	[	0.25
Species 2	[	]	[	]	0.25	0.5

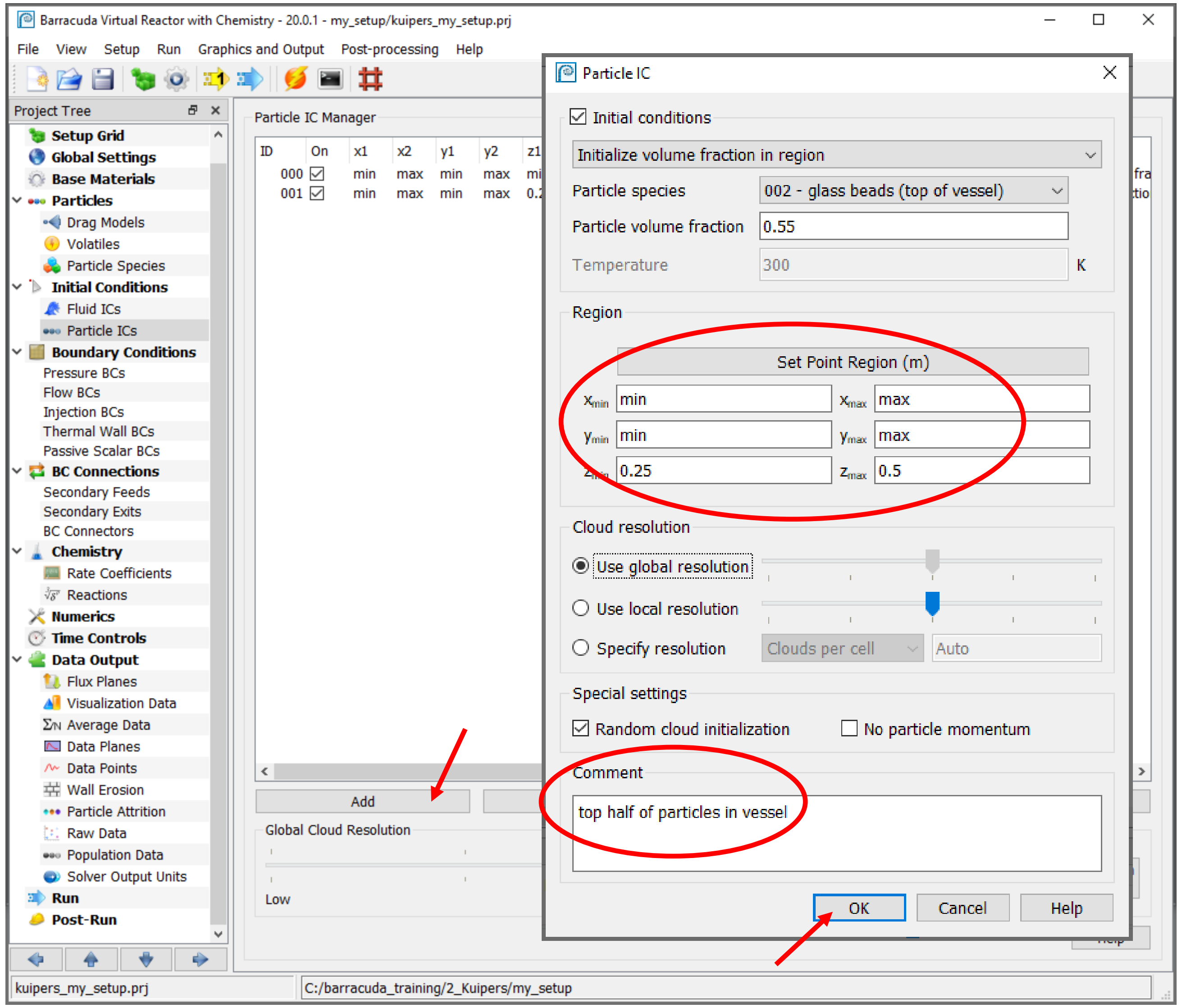


Particle Initial Conditions

Next, define the initial conditions for the second particle species:

- Click Add, which pops up a Particle IC dialog
- Select Initialize volume fraction in region
- Select 002 – glass beads in Particle Species
- Set the Particle volume fraction to 0.55
- Set Point Region as shown in the table below. [= min,] = max
- Add descriptive comment
- Click OK

Boundary	x		y		z	
Species 1	[	]	[	]	[	0.25
Species 2	[	]	[	]	0.25	0.5



Pressure Boundary Condition

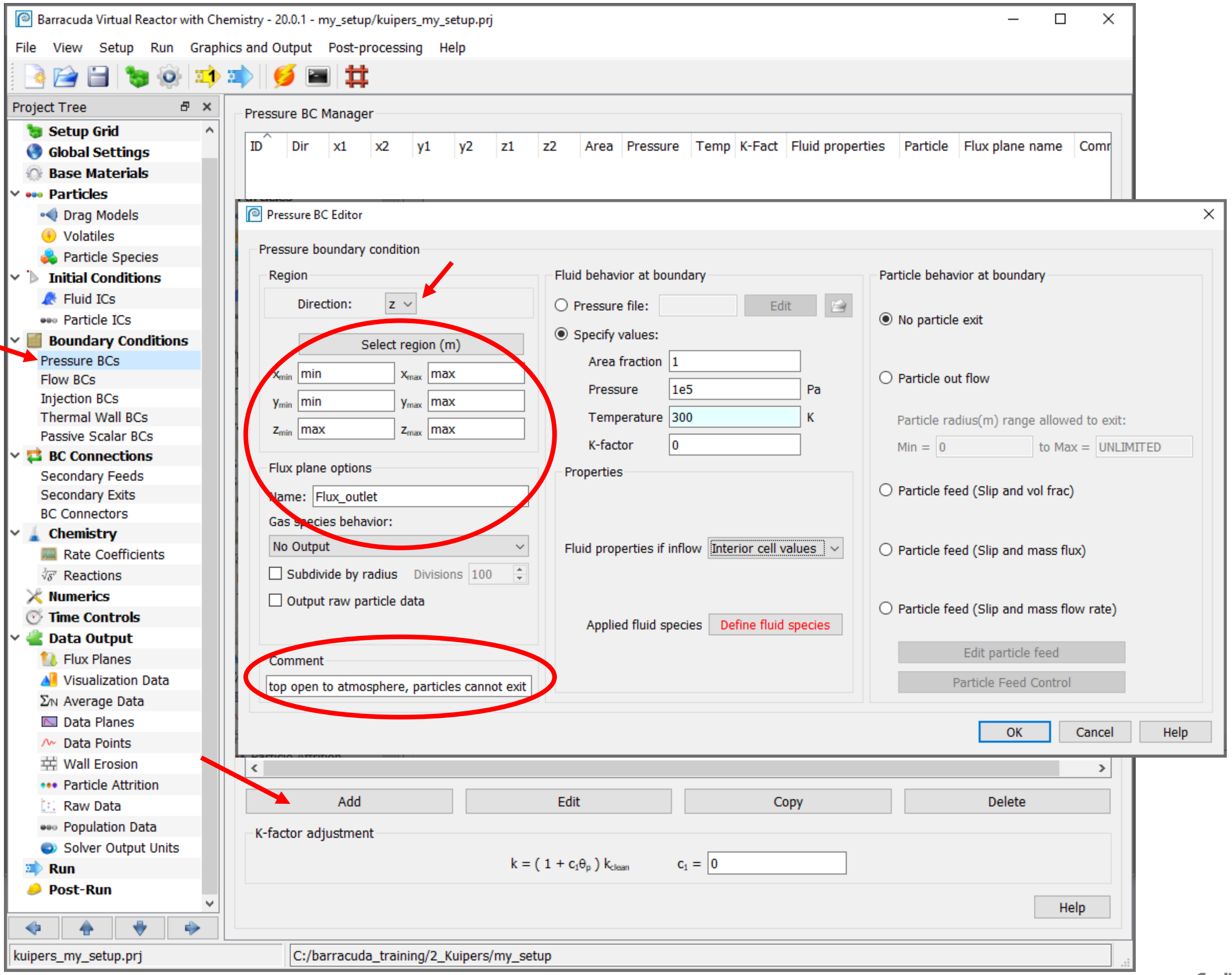
To create the top open to atmosphere:

- Click on Pressure BCs
- Click Add

In the Pressure BC Editor:

- Set Direction to z
- Set region using [and] (for min and max)
- Provide a name for the Flux plane
  - Best practice is to start a flux plane file name with **Flux_** or **FLUX_** for ease of post-processing
- Enter a comment (optional)

Boundary	x		y		z	
Fluidizing gas	[	-0.0075	[	]	[	[
Fluidizing gas	0.0075	]	[	]	[	[
Center jet	-0.0075	0.0075	[	]	[	[
Outlet Pressure	[	]	[	]	]	]



Pressure Boundary Condition

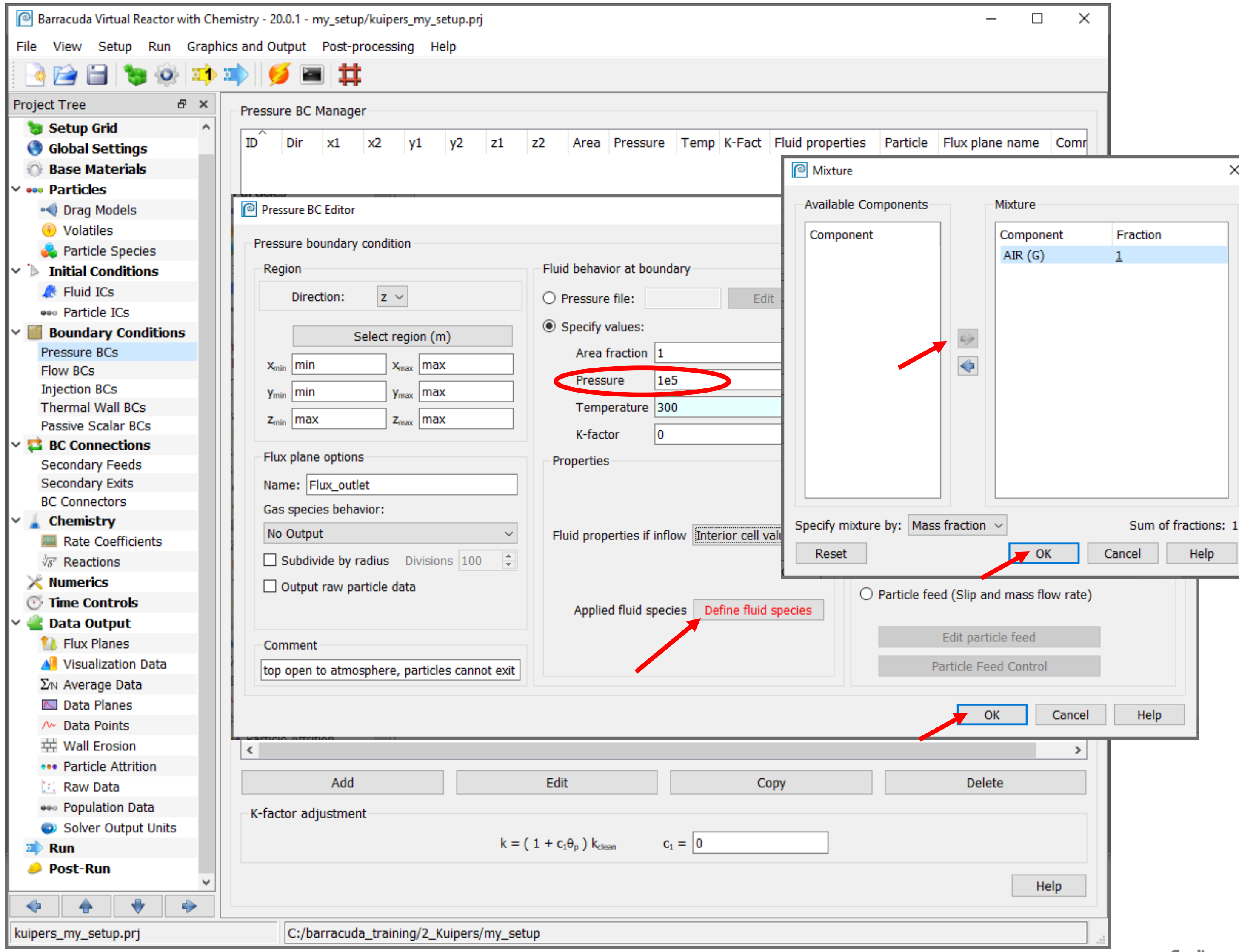
In the Pressure BC Editor:

- Specify the pressure as 1e5 Pa
- Click on Define fluid species

In the Mixture Editor:

- Import AIR(G) from Available Component to Mixture using the Right Arrow Button
- Click OK

Click OK in the Pressure BC Editor



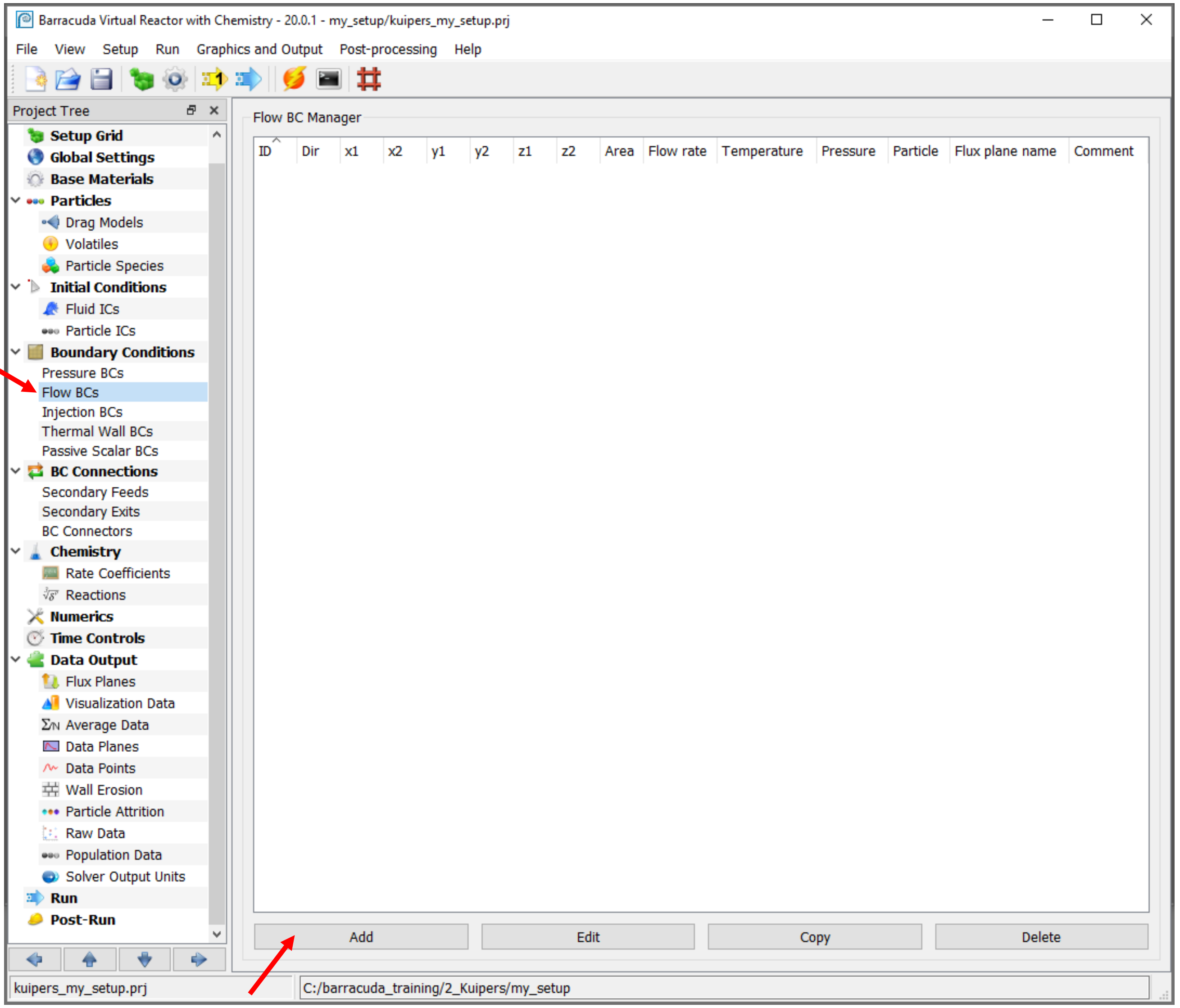
Flow Boundary Conditions

Click on Flow BCs

- The Flow BC page is used to define fluid boundary conditions where flow rate (mass or velocity) is known
- Flow rate can either be constant (using the Specify values option) OR entered as a transient value in a separate data file
- Each flow boundary (2 fluidizing gas and one center jet) has to be defined separately

Boundary	x		y		z	
Fluidizing gas	[	-0.0075	[	]	[	[
Fluidizing gas	0.0075	]	[	]	[	[
Center jet	-0.0075	0.0075	[	]	[	[
Outlet Pressure	[	]	[	]	]	]

Click on Add

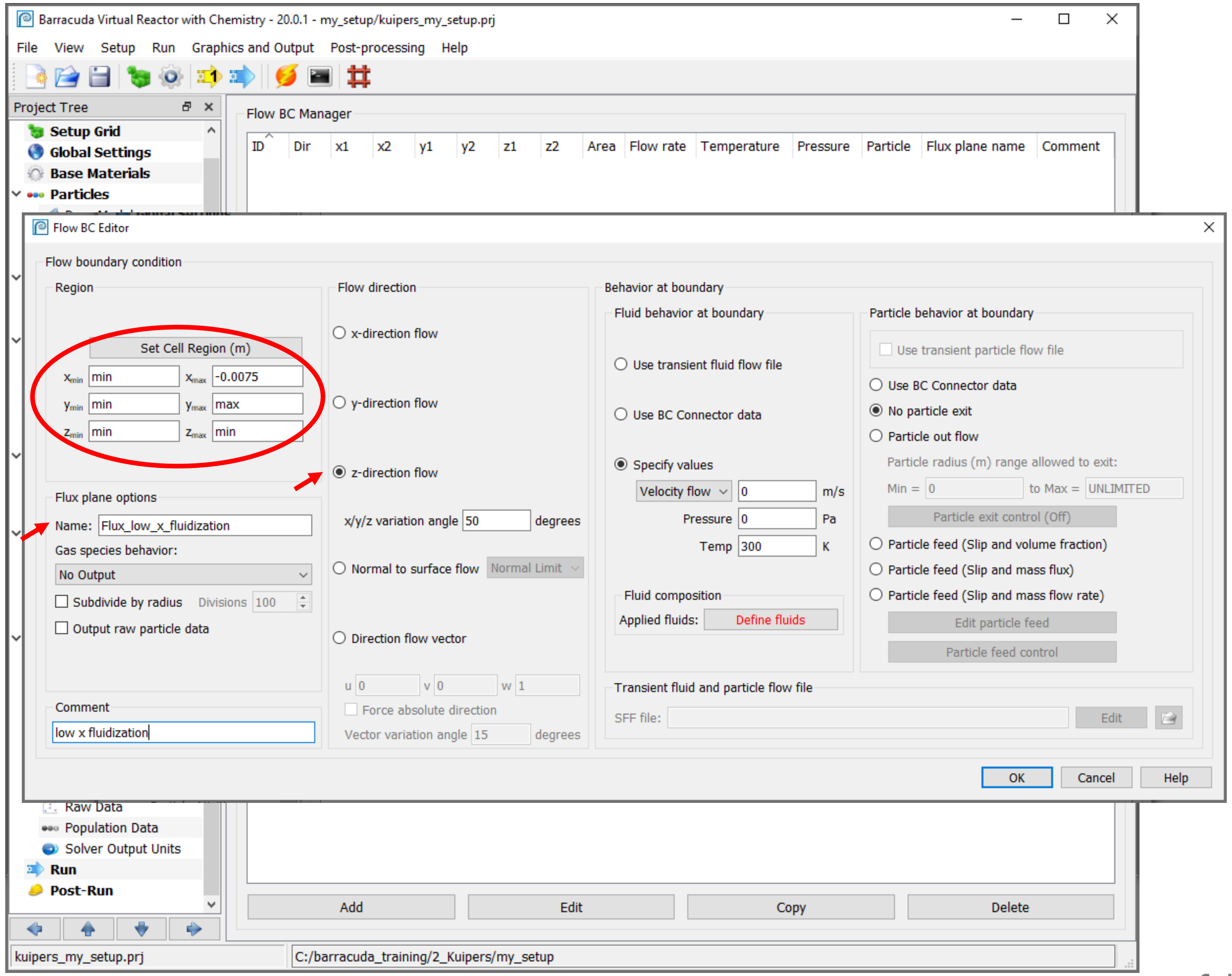


Flow Boundary Conditions – Low x Fluidizing Gas

Begin by defining the boundary for fluidizing air entering at low indices in the x-direction:

- Specify the Cell Region as shown below
- Provide a Flux plane name for the boundary
- Enter a Comment (optional)
- Leave the Flow direction setting at z-direction

Boundary	x		y		z	
Fluidizing gas	[	-0.0075	[	]	[	[
Fluidizing gas	0.0075	]	[	]	[	[
Center jet	-0.0075	0.0075	[	]	[	[
Outlet Pressure	[	]	[	]	]	]



Flow Boundary Conditions

Specify values:

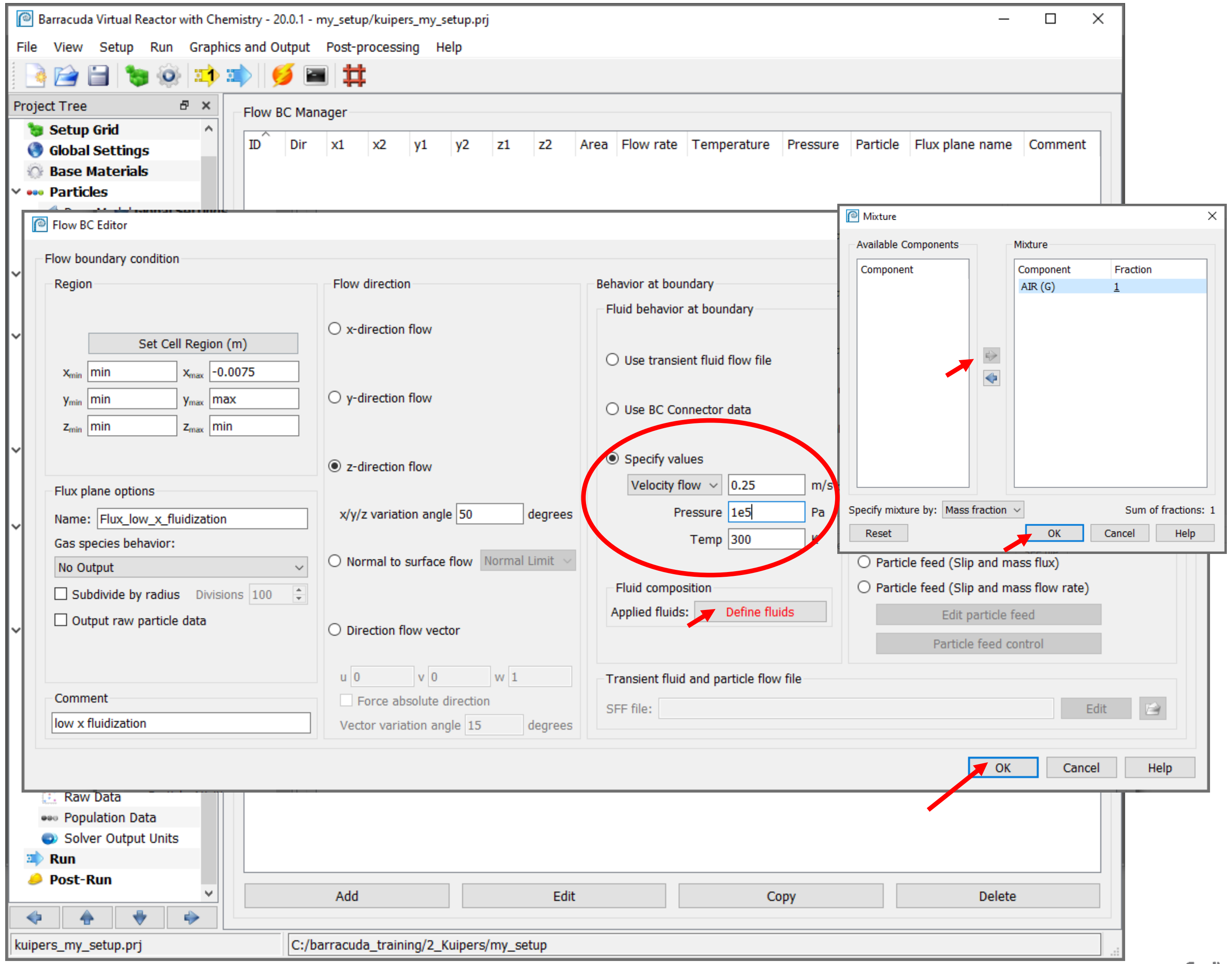
- Set Velocity flow to 0.25 m/s
  - A positive velocity means flow is directed into the model space (negative means out-flow)
  - The velocity is a superficial velocity through an open area, in the absence of particles
- Set Pressure to 1e5 Pa

Click on Define fluids

Mixture:

- Select AIR (G)
- Use the Right arrow button to import from Available Components to Mixture
- Click OK

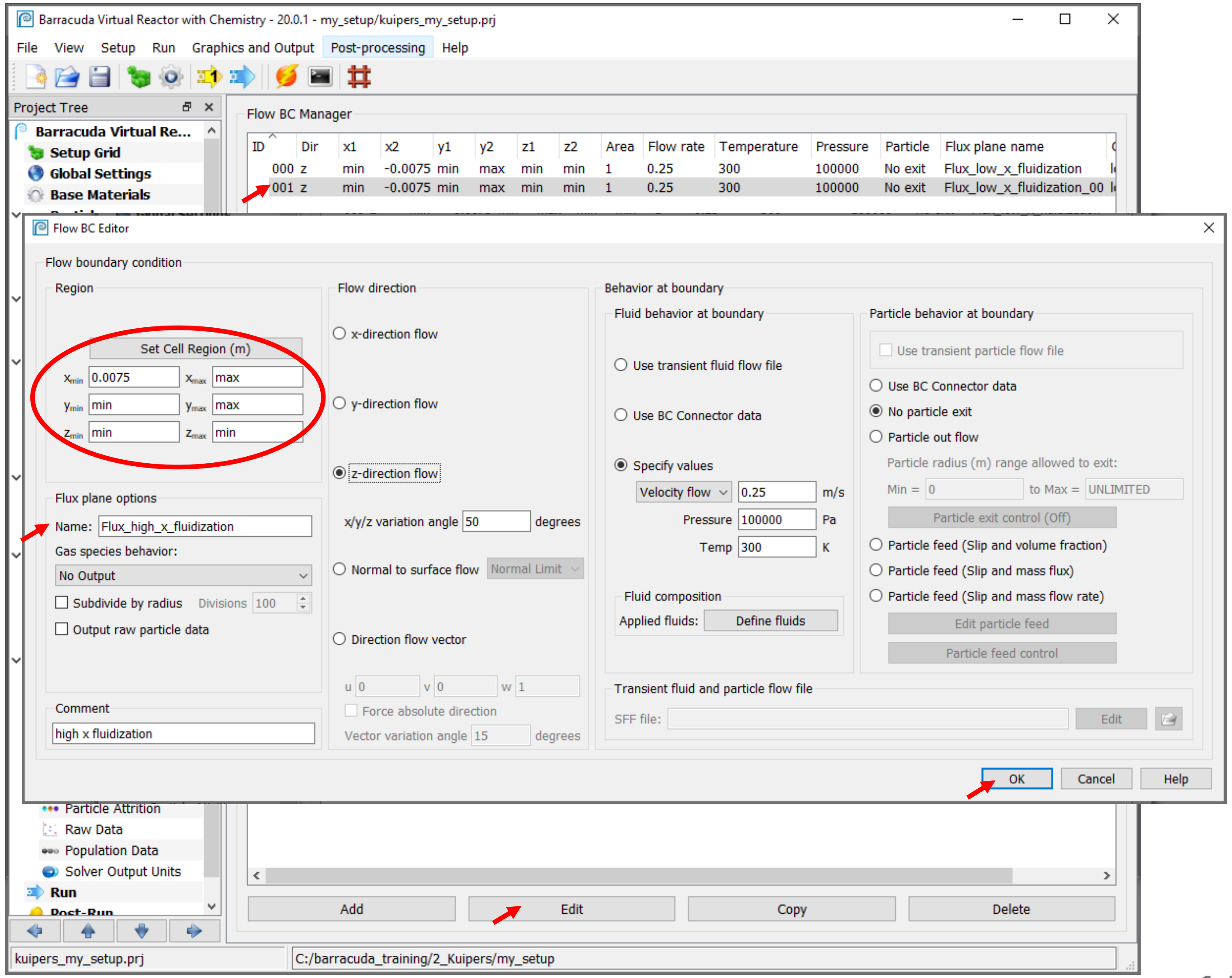
Click OK in Flow BC Editor



Flow Boundary Conditions - High x Fluidizing Gas

Create the boundary for fluidizing air entering at high indices in x-direction:

- Copy the first Flow BC
- Select the copy and click on Edit
- Edit the Cell Region as shown below
- Edit the Flux plane name to correspond with the boundary
- Enter a Comment (optional)
- Click OK

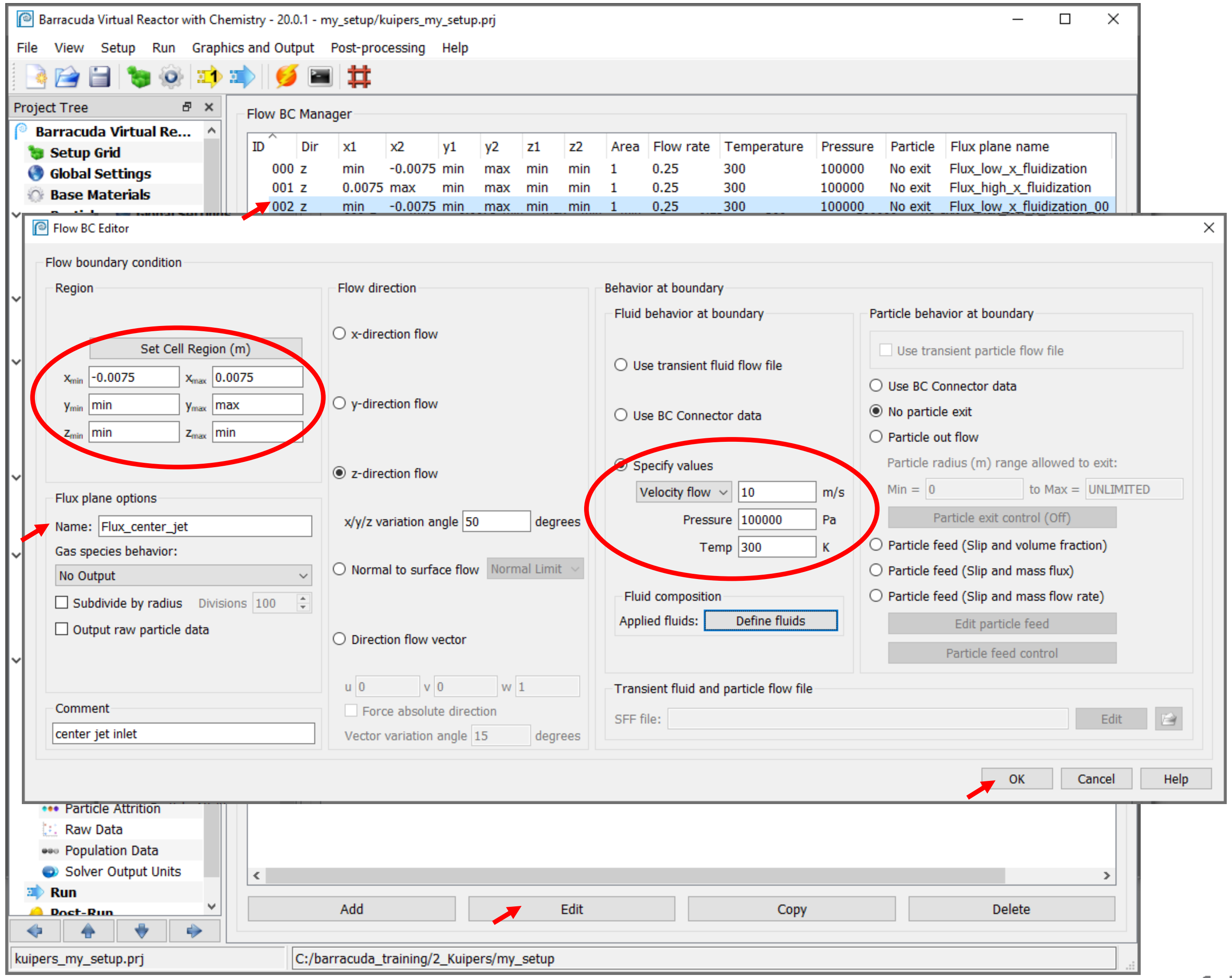


Boundary	x		y		z	
Fluidizing gas	[	-0.0075	[	]	[	[
Fluidizing gas	0.0075	]	[	]	[	[
Center jet	-0.0075	0.0075	[	]	[	[
Outlet Pressure	[	]	[	]	]	]

Flow Boundary Conditions – Center Jet

Create the boundary for center jet:

- Copy the first Flow BC
- Select the copy and click on Edit
- Edit the Cell Region as shown below
- Edit the Flux plane name to correspond with the boundary
- Enter a Comment (optional)
- Set Velocity flow to 10 m/s
- Click OK



Boundary	x		y		z	
Fluidizing gas	[	-0.0075	[	]	[	[
Fluidizing gas	0.0075	]	[	]	[	[
Center jet	-0.0075	0.0075	[	]	[	[
Outlet Pressure	[	]	[	]	]	]

Time Controls

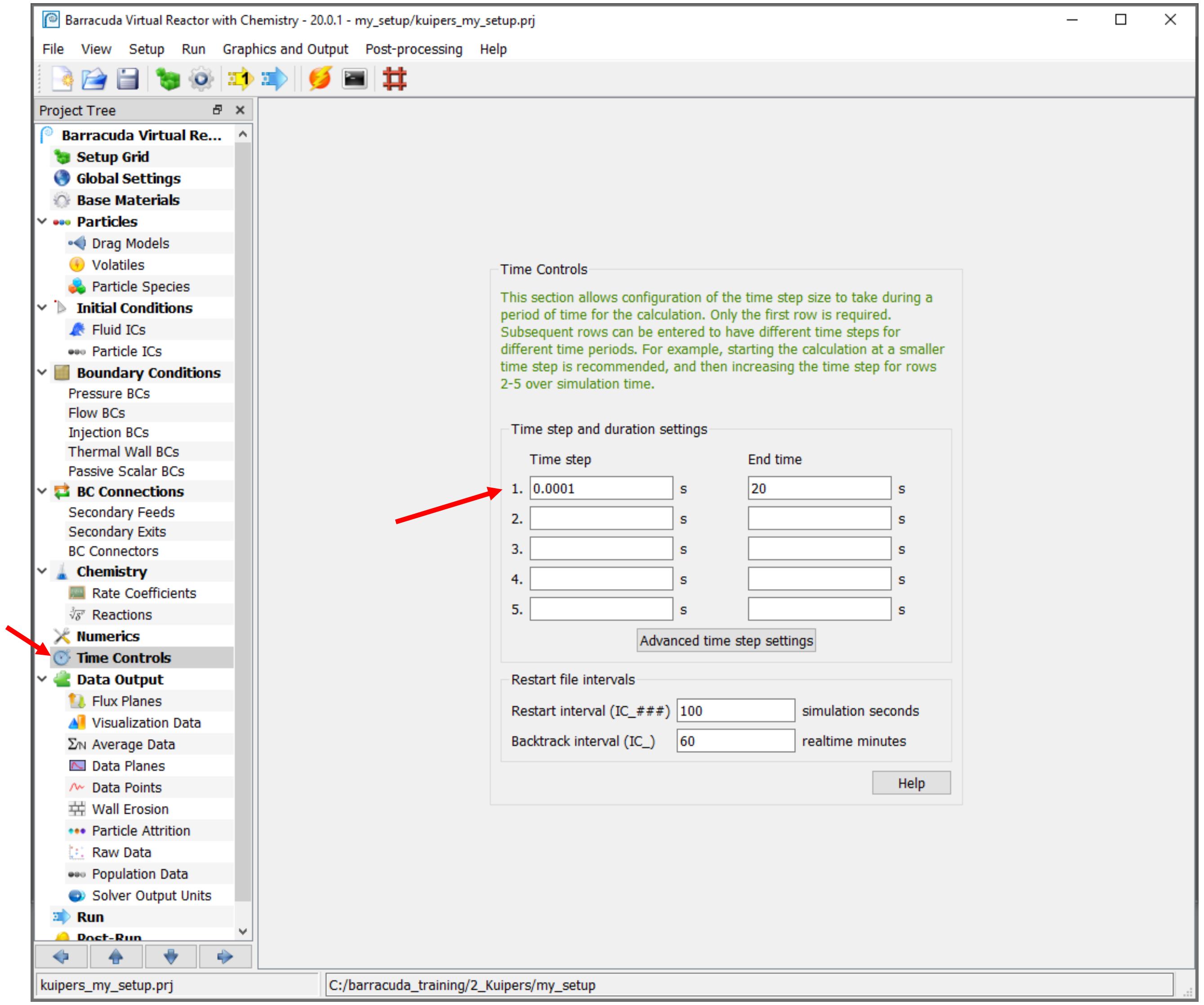
Click on Time Controls:

- Leave the Time step as 0.0001 s
- Set the End time to 20 s

Barracuda can restart an existing simulation from an IC file. Two types of IC files are automatically written during a simulation:

- Restart file: a restart IC file is written once at every specified interval of simulation time
- Backtrack file: a backtrack IC file is written once at every specified interval of clock time

Leave the restart file intervals at the default values

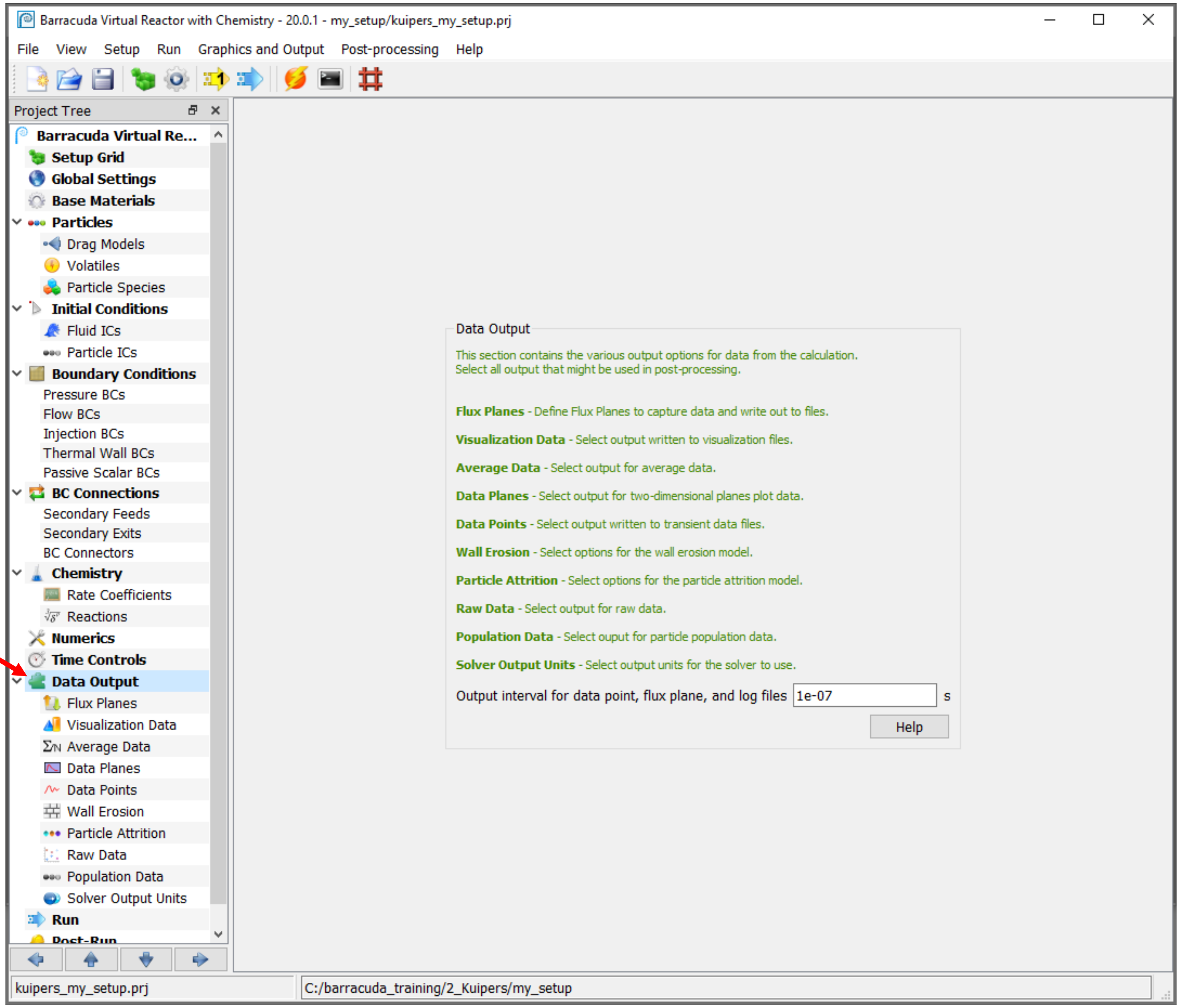


Data Output Options

Click on Data Output

For the Kuipers problem, the following types of data will be output:

- Flux planes – Track the transport of fluid and solids through a defined plane in the model
- Visualization Data – Select variables for visualization of fluid and solid states in Tecplot
- Average Data – Select some Tecplot output data to be averaged as the simulation runs
- Data Points – High frequency tracking of data at a specified location in a model

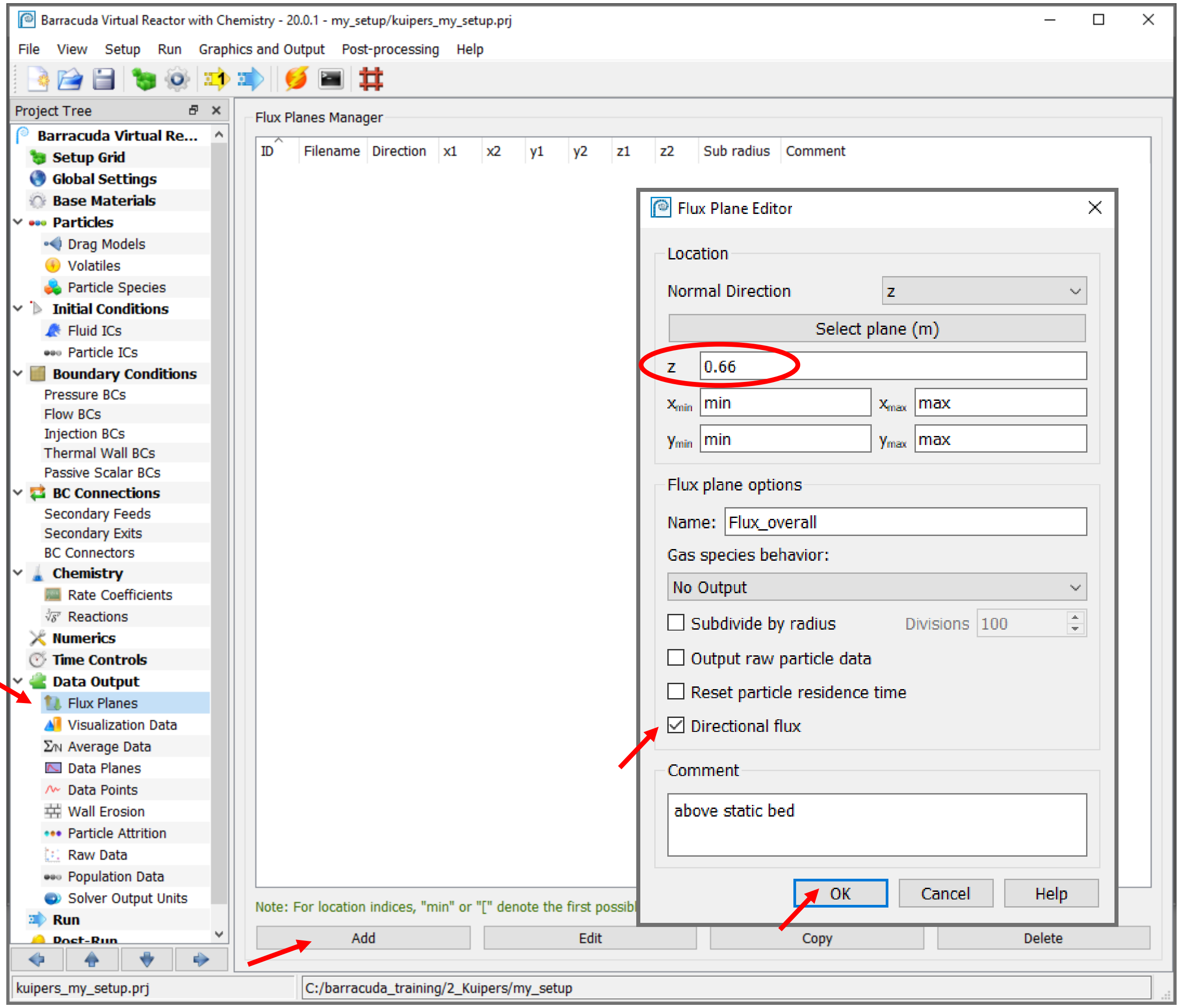


Data Output: Flux Planes

Click on Flux Planes

To create interior flux planes:

- Click Add
- Select z for the Normal Direction
- Enter the z location as 0.66 m
- Provide a descriptive Name for the flux plane
- Select Directional flux
- Enter a Comment (optional)
- Click OK



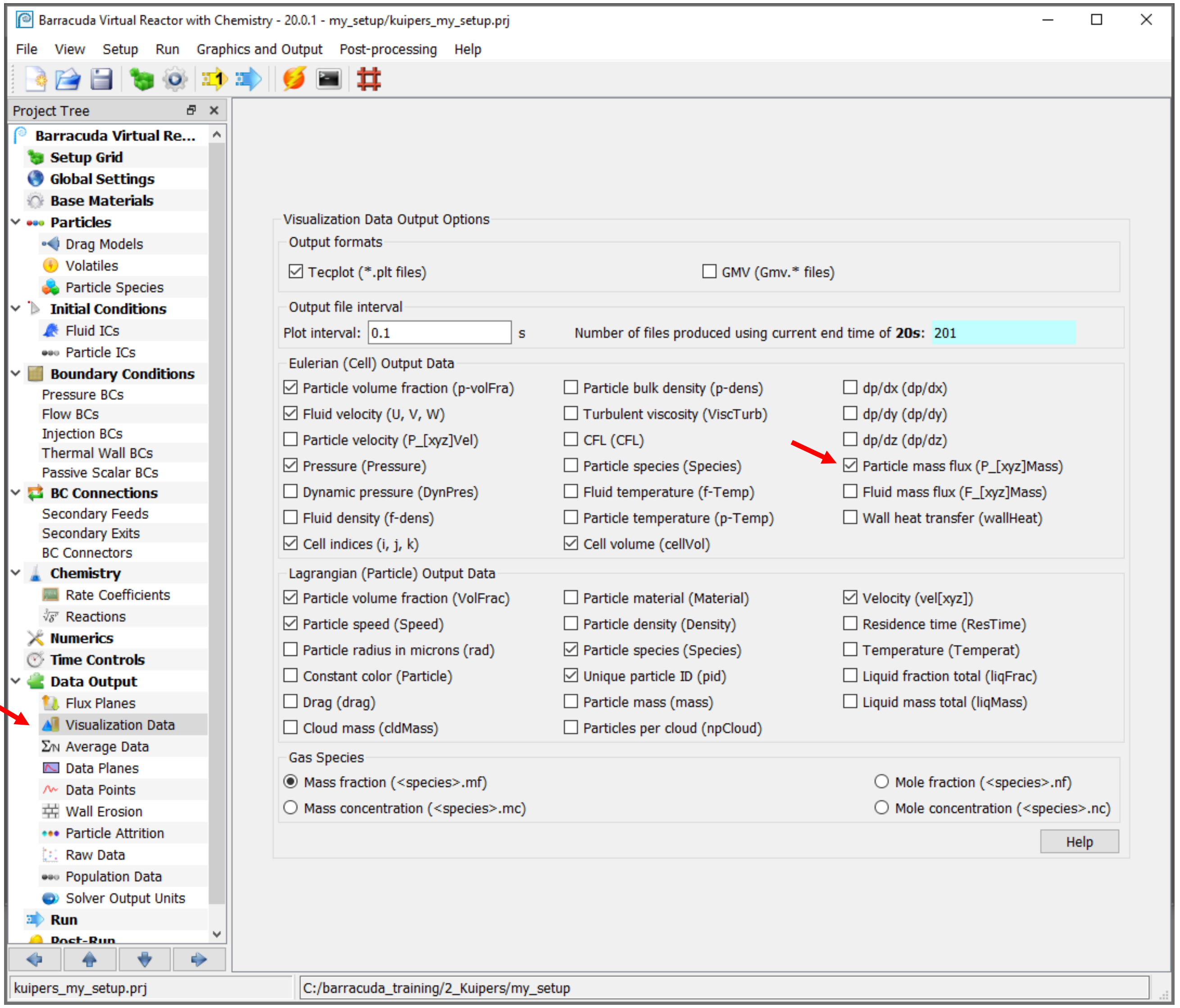
Data Output: Visualization Data

Click on Visualization Data

The information selected here will be written to your Tecplot output files

- Eulerian Output Data is mapped to the grid
- Lagrangian Output Data is mapped to particle locations

Select Particle mass flux (P_[xyz]Mass) in addition to default selections



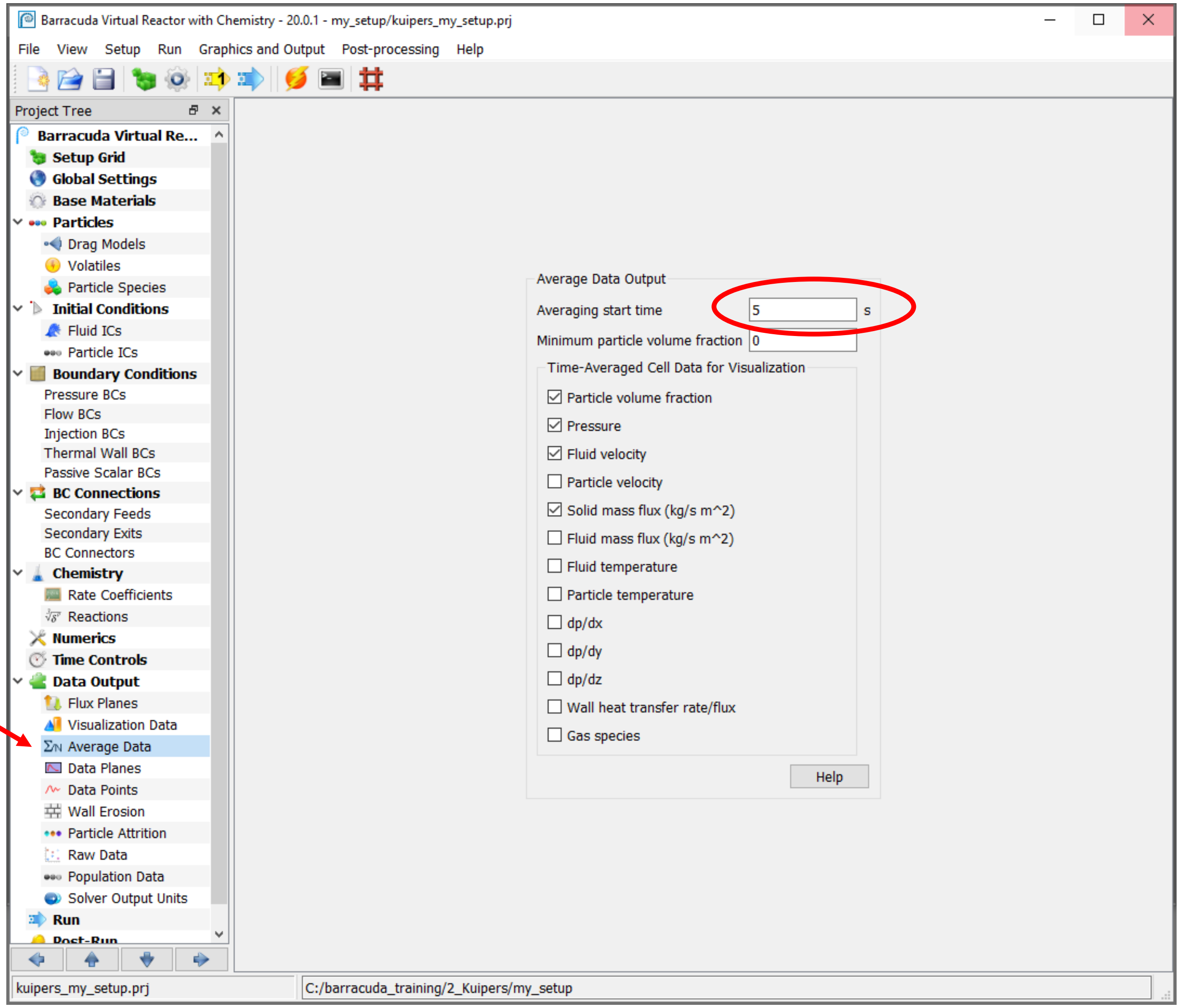
Data Output: Average Data

Often time-averaged data gives more insight into quasi-steady behavior than instantaneous data

- Select a Start time for average once you think quasi-steady behavior will have begun
- Select the data you wish to average

Click on Average Data

- Set Averaging start time to 5 s
- Select the following options for time-averaging:
  - Particle Volume Fraction
  - Pressure
  - Fluid velocity
  - Solid mass flux



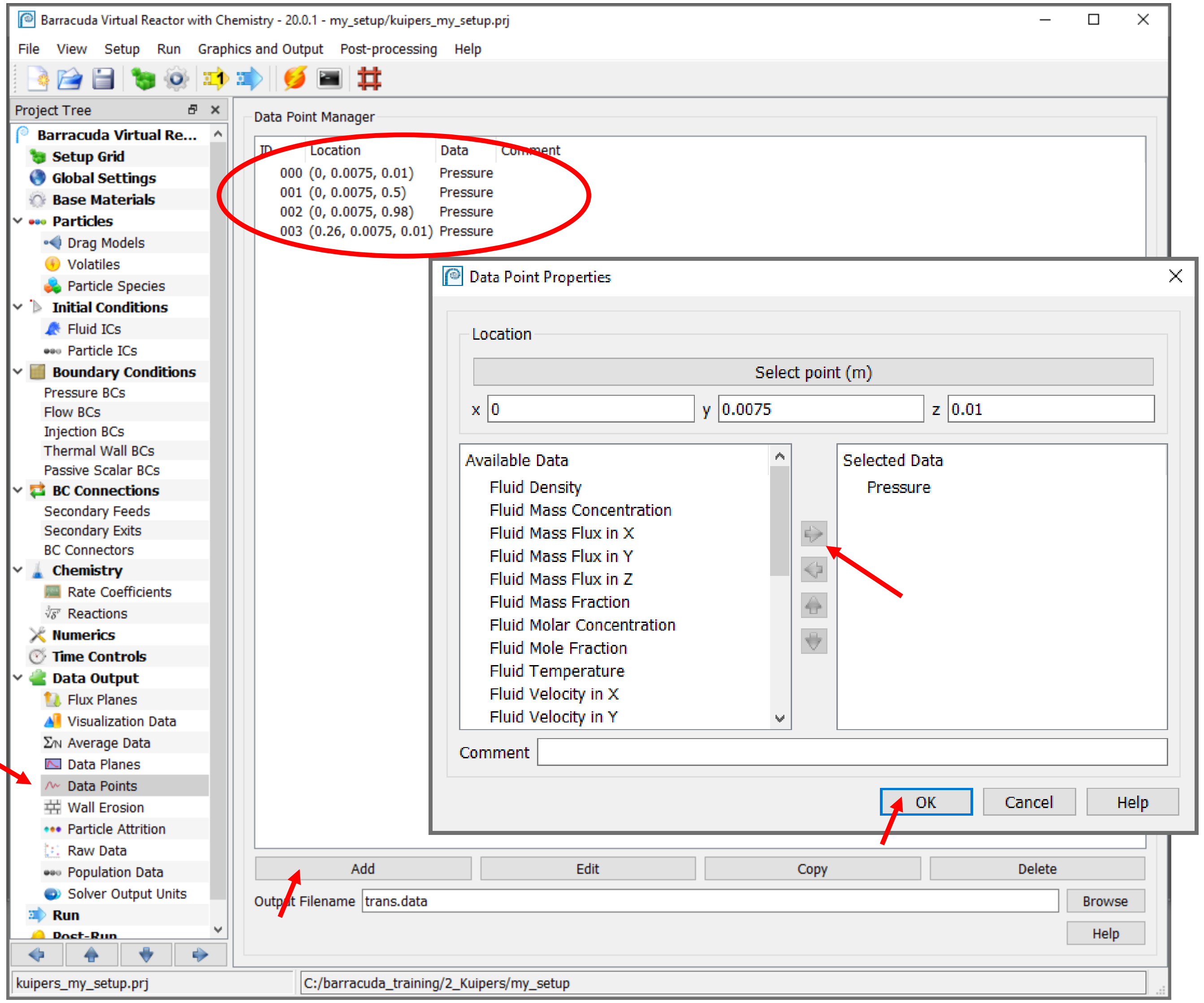
Data Output: Data Points

Barracuda can output high-frequency data at point locations in the computational domain

Data will be written to the file: `trans.data00`

To create the four data points used in post-processing:

- Click on Data Points
- Click on Add
- Enter the coordinates shown in the Location column for one Data Point
- Import Pressure from Available Data to Selected Data
- Click OK
- Repeat for additional three data points using the corresponding location for each point



Run Solver

Run – GPU Parallel

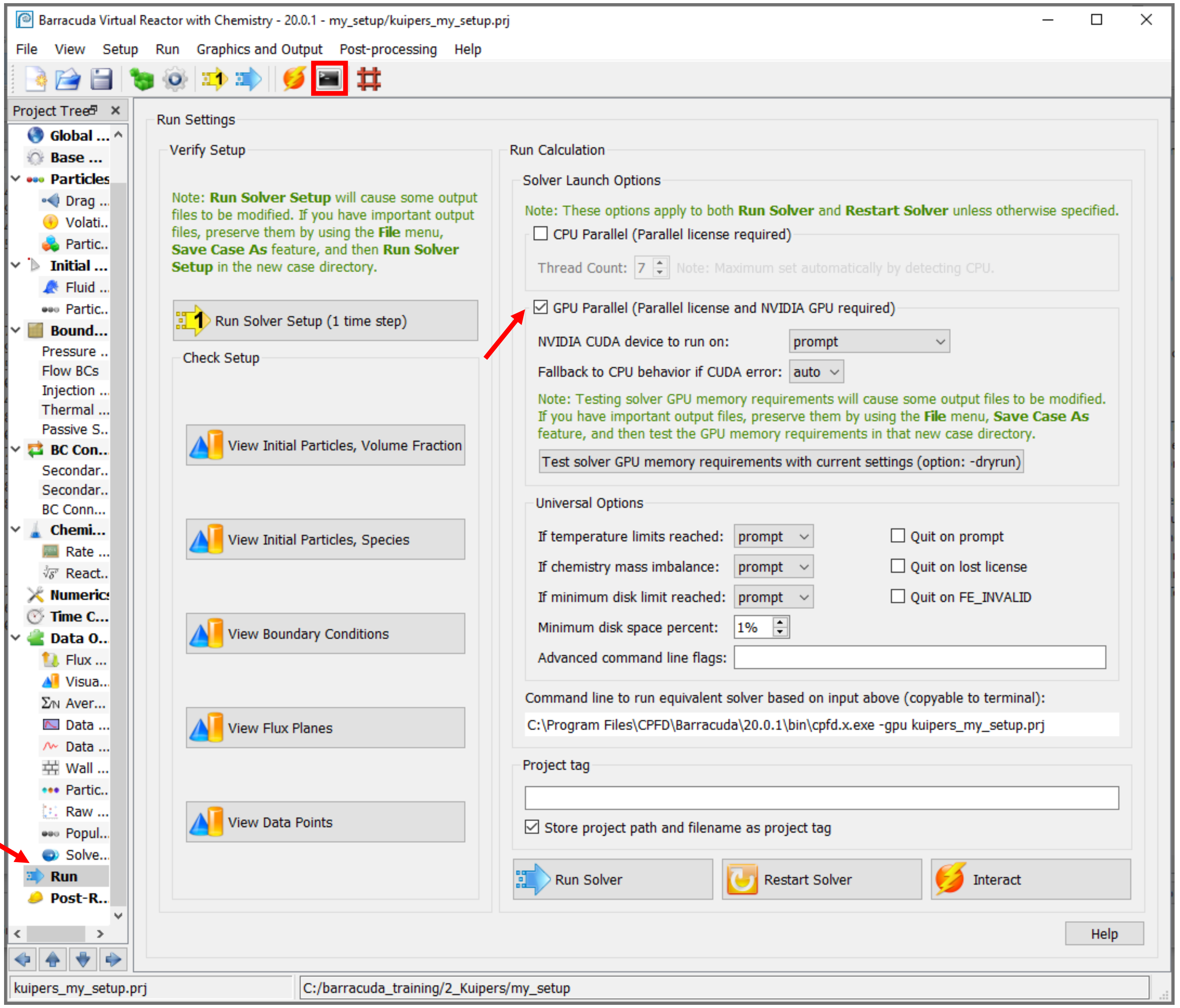
Click on Run

If your calculation machine has an NVIDIA GPU, select GPU Parallel

Check your GPU configuration by clicking on the command window and typing in the following prompt:

- Windows: `C:\Program Files\NVIDIA Corporation\NVSMI\nvidia-smi.exe`
- Linux: `nvidia-smi`

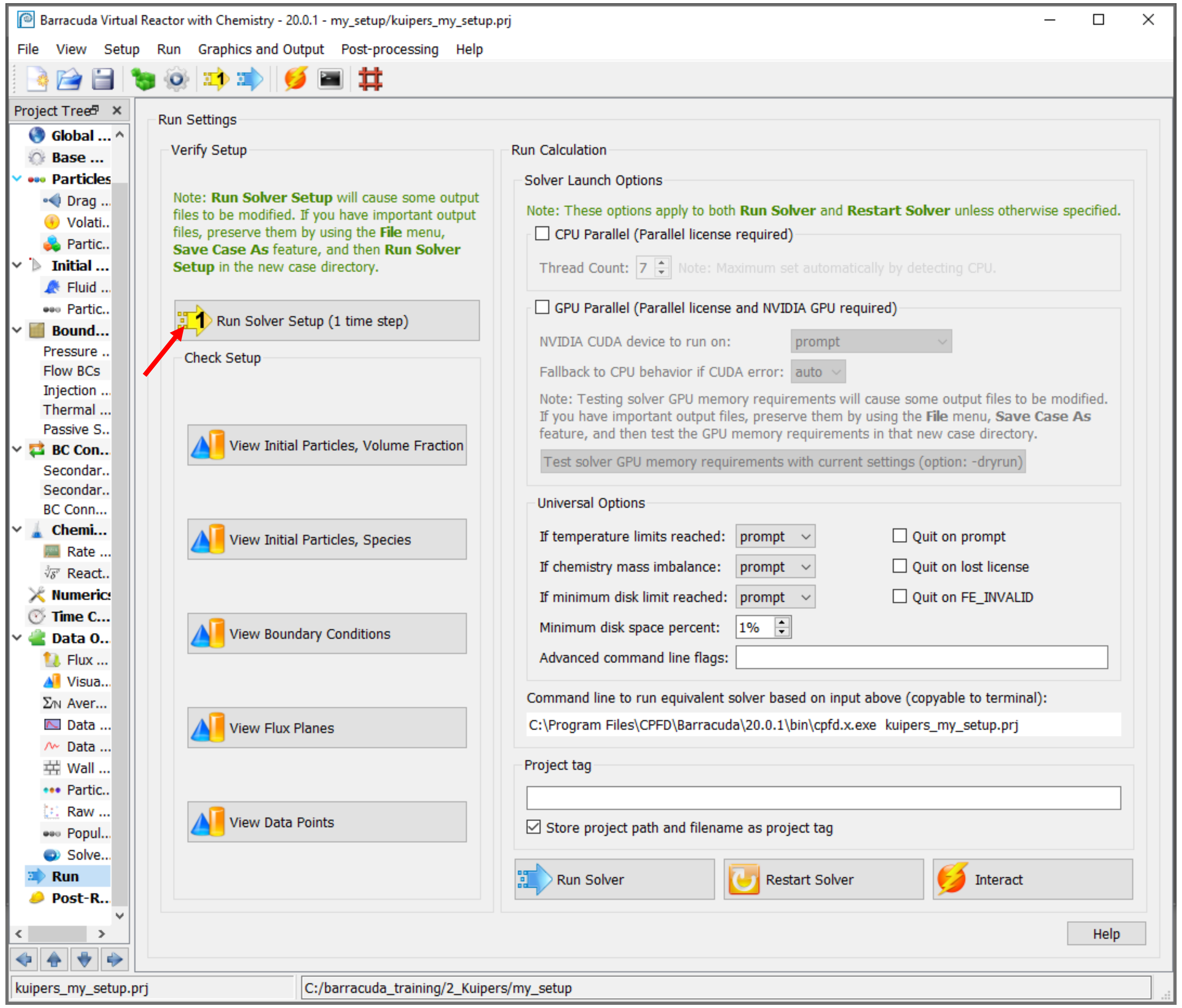
See this support site post for more information.



Run Solver Setup

Click on Run Solver Setup

- This automatically saves the project file
- This will run the simulation for one time step and write a single Tecplot file
- All boundary and initial conditions are stored in the Tecplot file
- This first Tecplot file is needed to check the problem setup

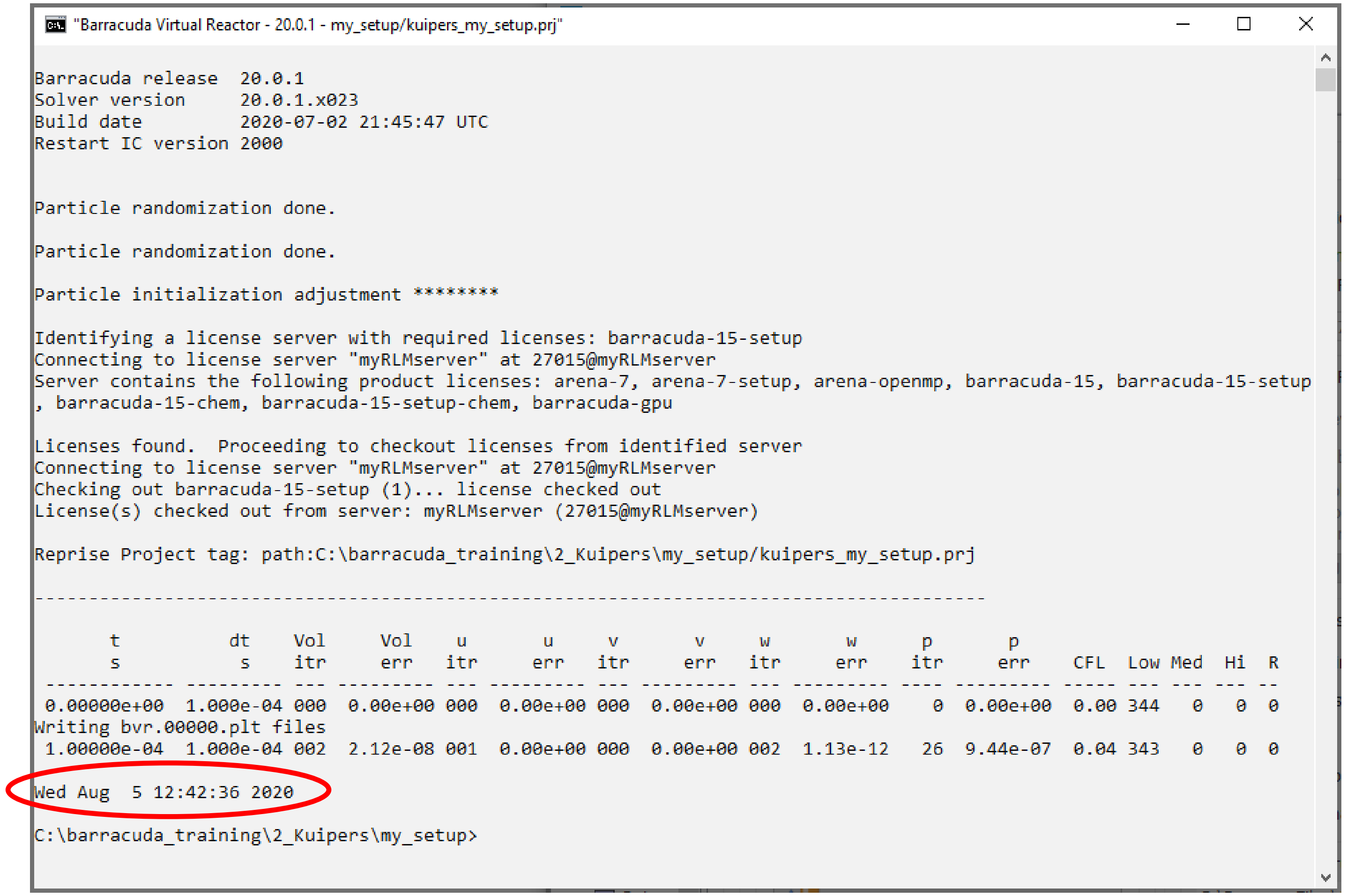


Run Solver Setup

Solver information will be output to the run window

Leave the window open while a calculation is running. If it is closed, the calculation will stop!

The calculation is complete when the date/time stamp is displayed at the bottom of the solver window. Once this happens, then you can close the run window.



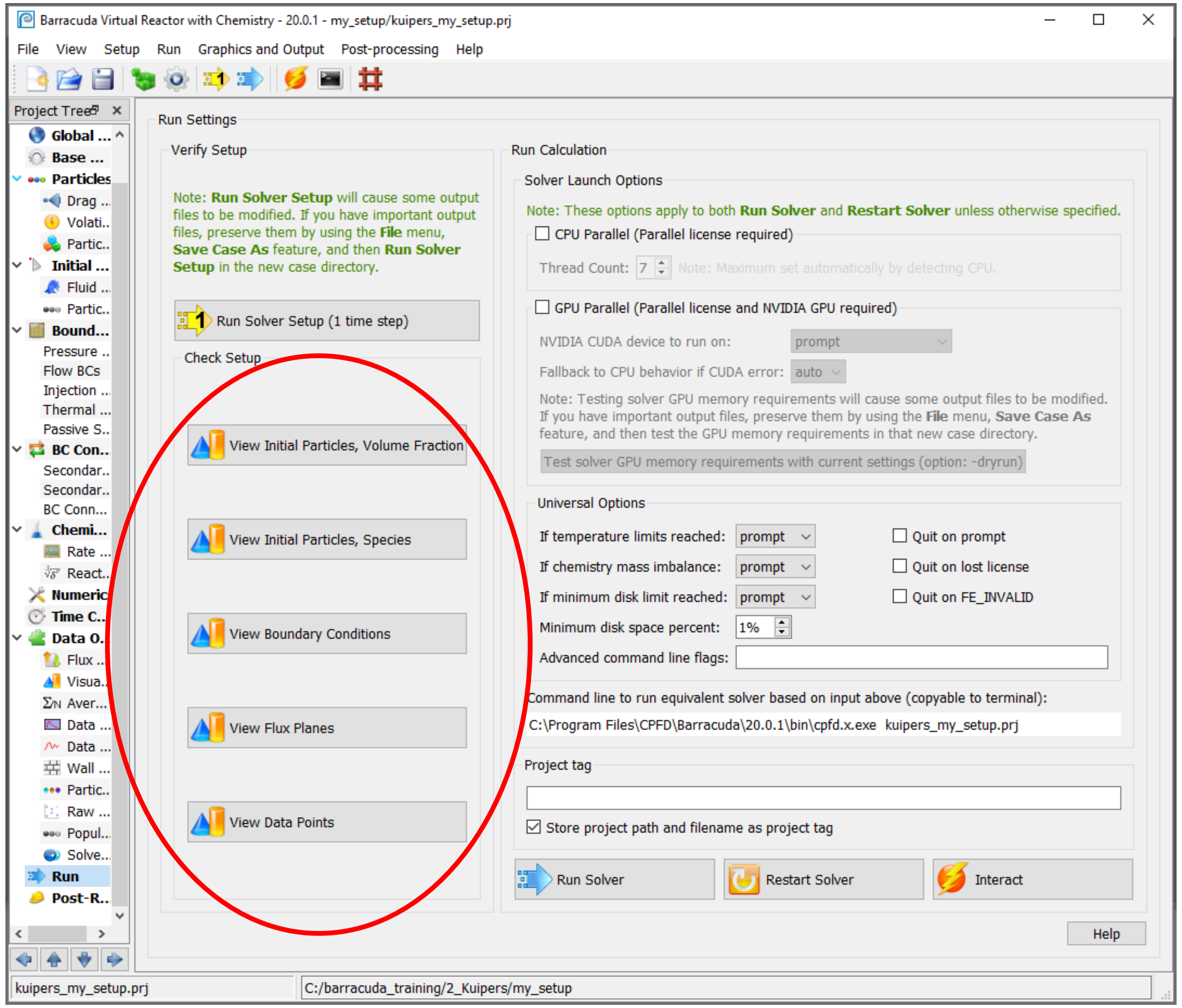
Checking your Setup

Before letting your calculation run, check the following:

- Are your boundary conditions in the right place?
- Are your particles defined correctly and located where you want them?
- are your data points and flux plane located where you want them?

To check, use the Check Setup buttons which open up corresponding Tecplot views.

- Watch our video on [Launching Tecplot]()



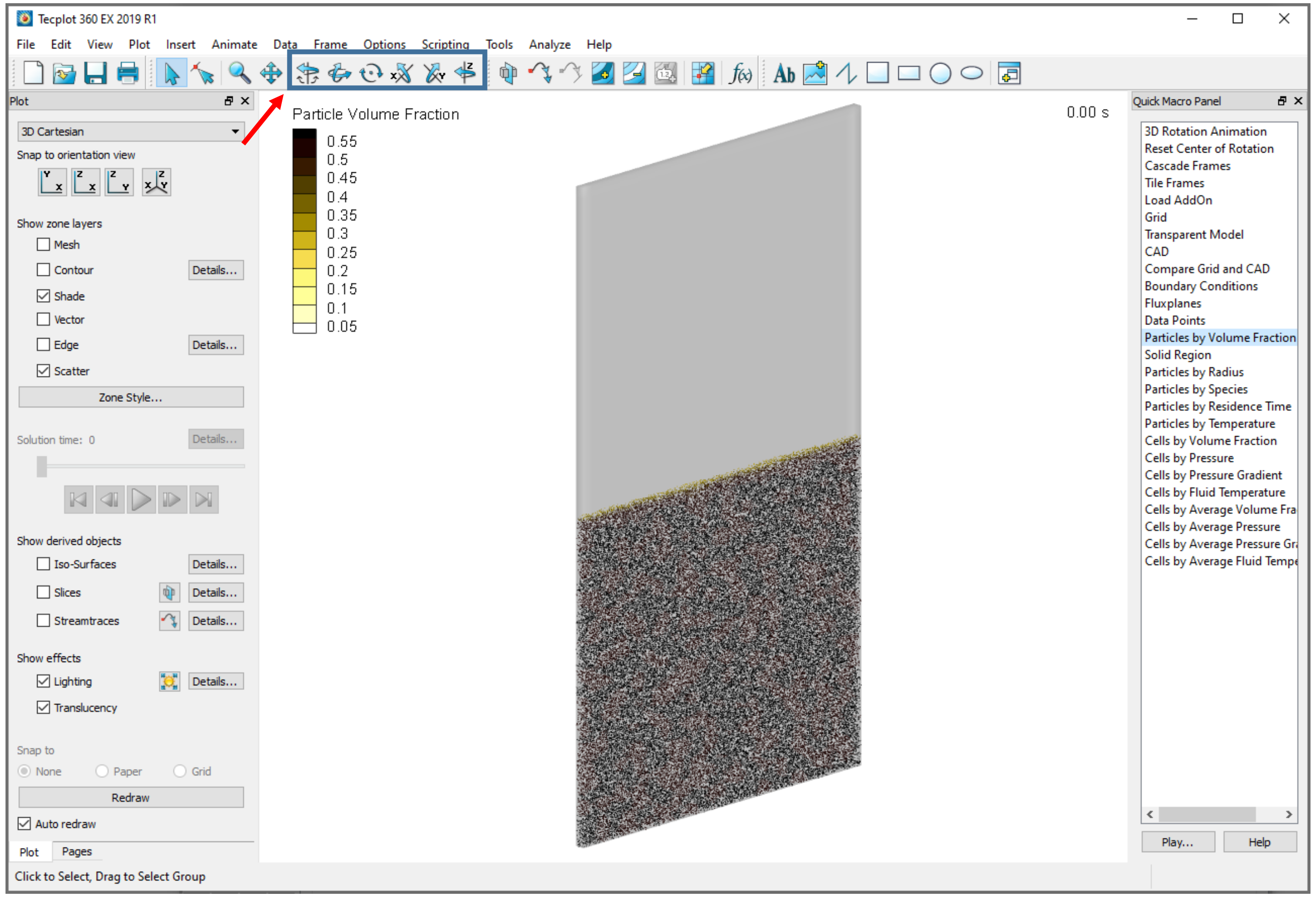
Checking your Setup - Particles

In the Run window, Click on View Initial Particles, Volume Fraction

Verify the initial location of the particles in the bed

To change the view, use the rotation tools in the toolbar

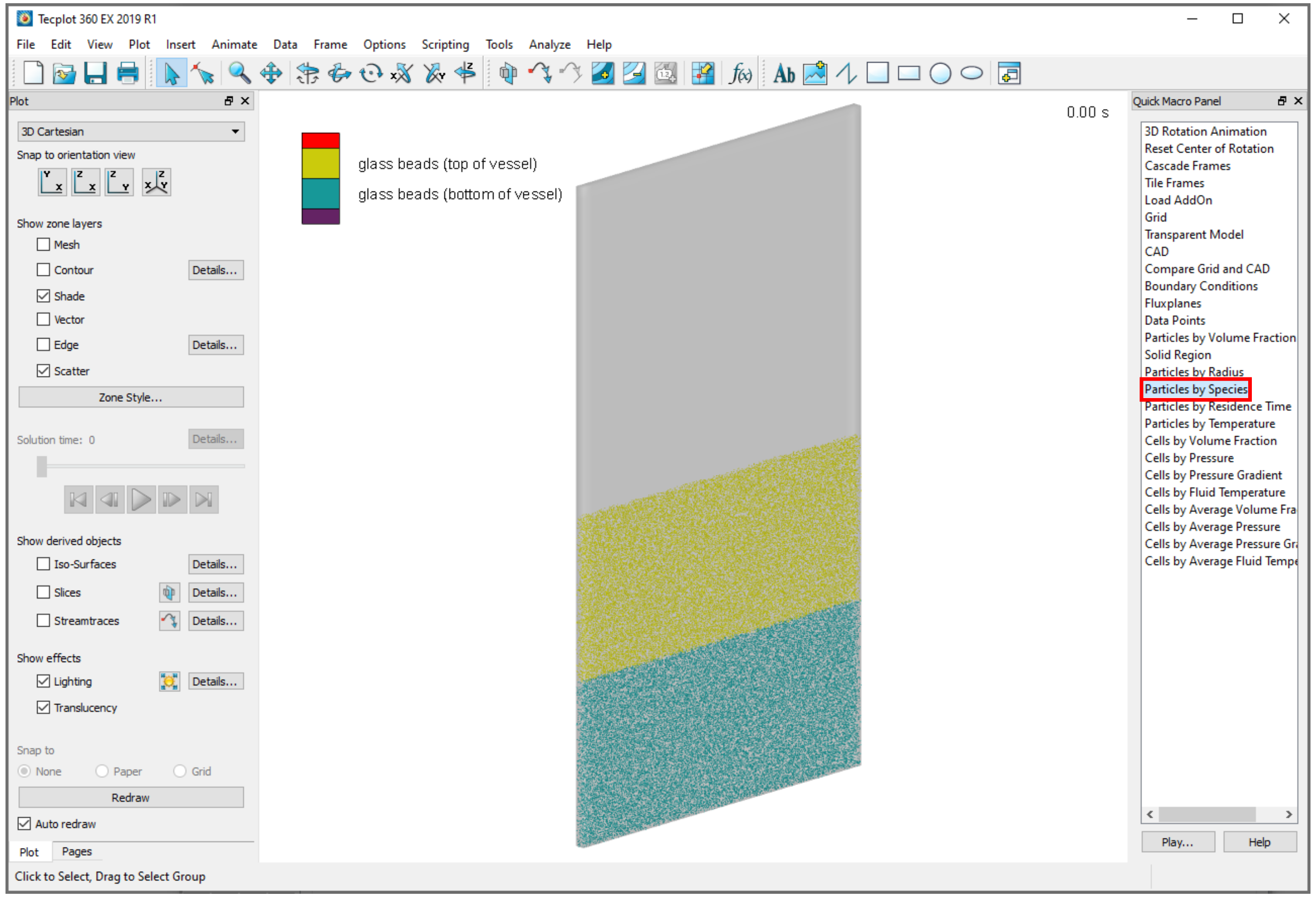
- Watch Tecplot's video [Tecplot 360 Basics: Rotate and Zoom]



Checking your Setup – Particle Species

Double-click on Particles by Species in the Quick Macro Panel. This corresponds to the view created by View Initial Particles, Species in the Run window.

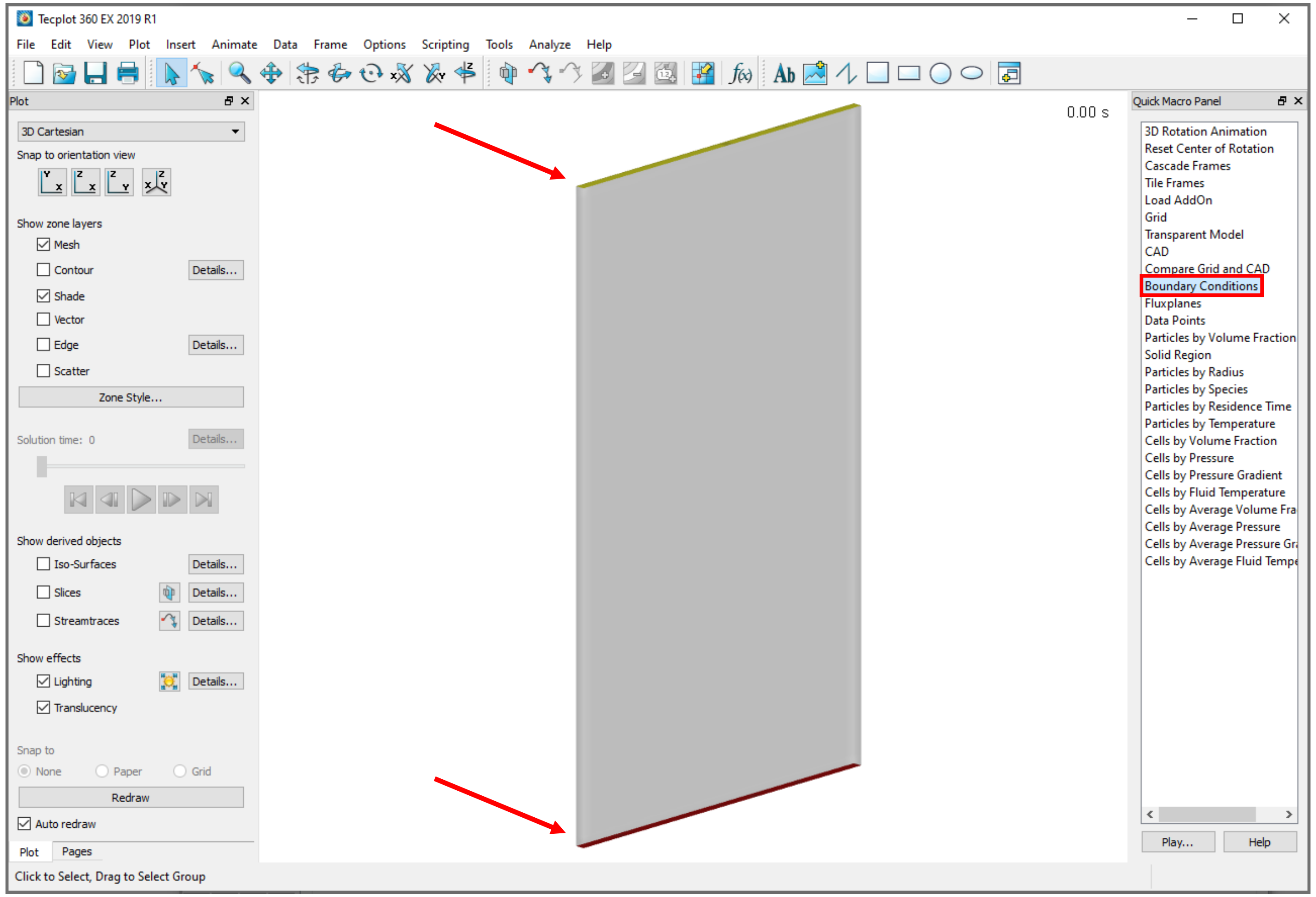
Verify the initial location of the two particle species in the bed



Checking your Setup – Boundary Conditions

Double-click on Boundary Conditions in the Quick Macro Panel. This corresponds to the view created by View Boundary Conditions in the Run window.

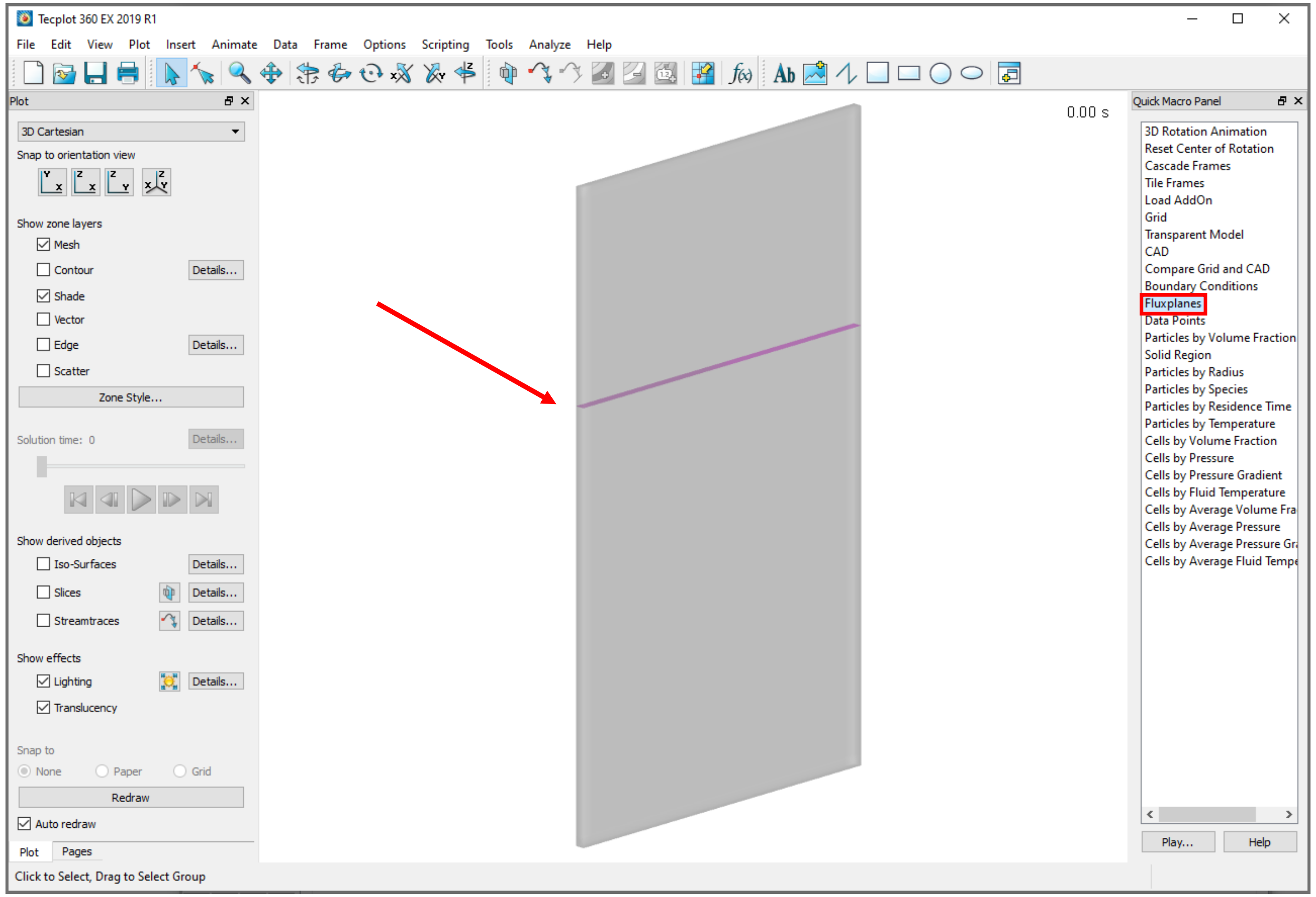
Verify that the boundary conditions are in the right place



Checking your Setup – Flux Plane Location

Double-click on Fluxplanes in the Quick Macro Panel. This corresponds to the view created by View Flux Planes in the Run window.

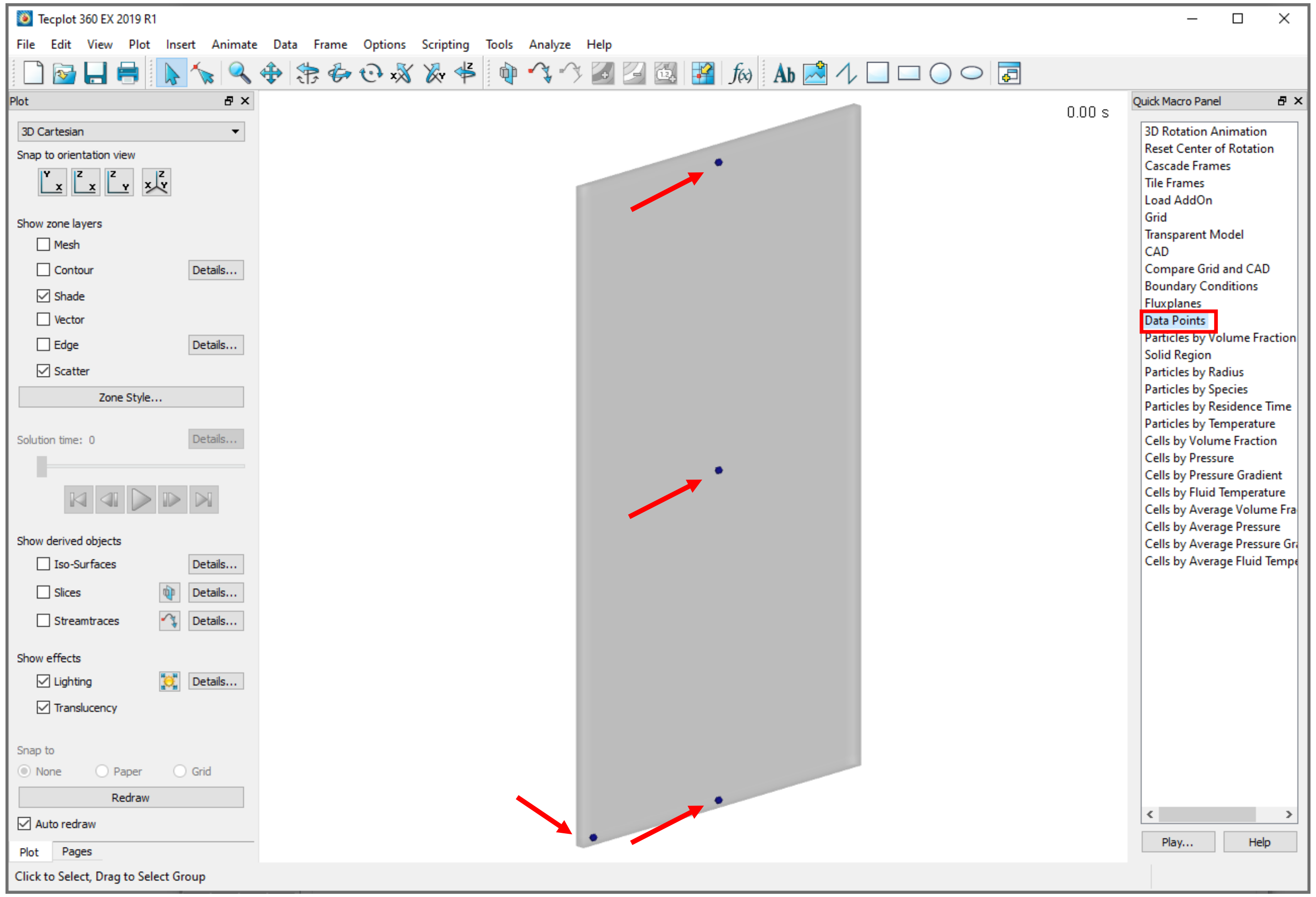
Verify that the flux plane is in the right place



Checking your Setup – Data Points Location

Double-click on Data Points in the Quick Macro Panel. This corresponds to the view created by View Data Points in the Run window.

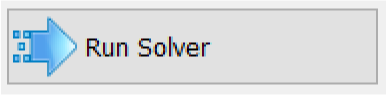
Verify that the data points are in the right place



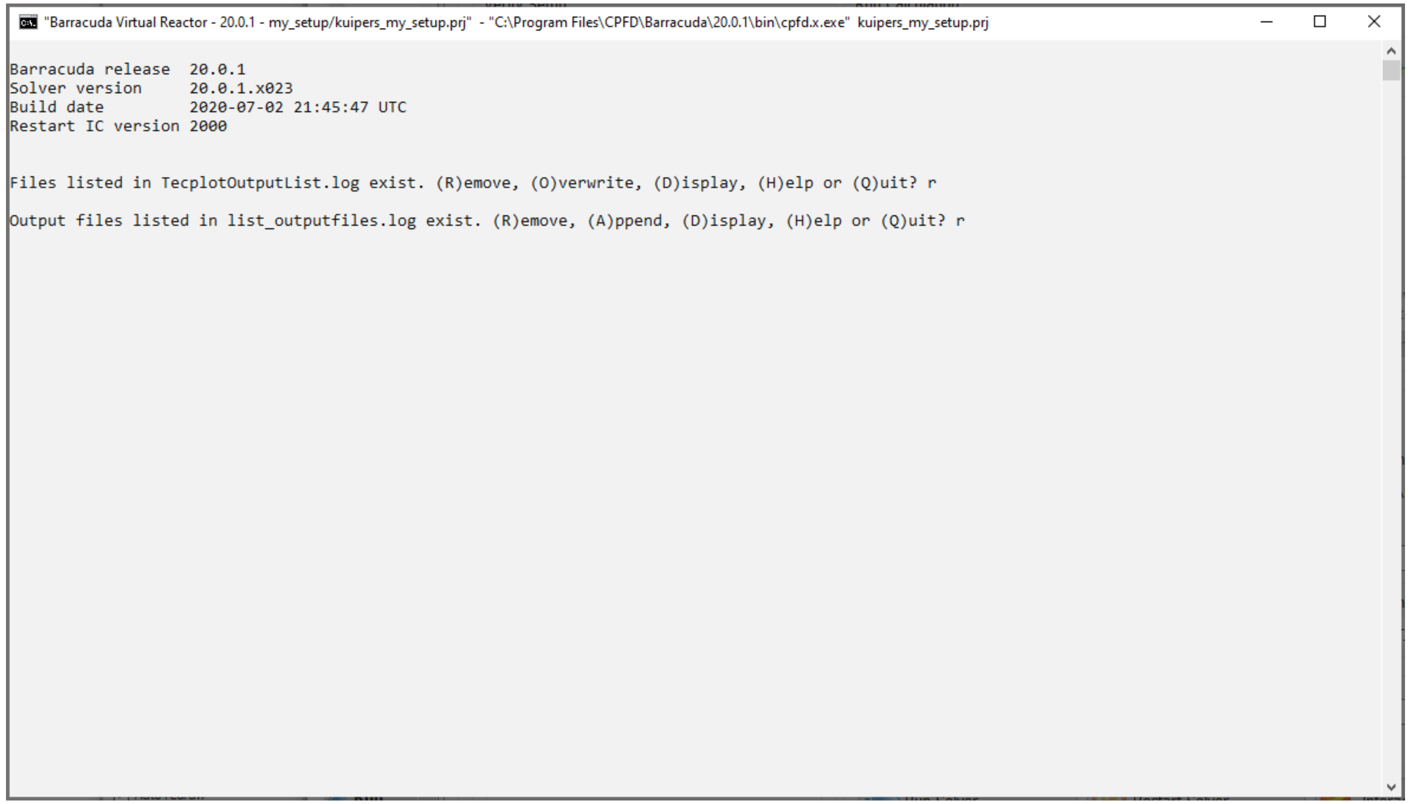
Executing the Simulation

STOP! Show your initial setup to your instructor before proceeding.

Once checked, click on Run Solver

 **or** 

The run window will prompt to Remove, Overwrite, Display, Help or Quit. The files can be removed since they were all created during the Run Solver Setup (1 time step).



Solver Output Window

Current Simulation Time

Current Time step

Solver Convergence Data

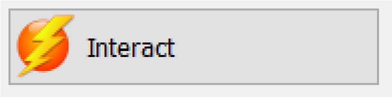
CFL number: Typically safe to run between 0.7 – 1.5

```
Select "Barracuda Virtual Reactor - 20.0.1 - my_setup/kuipers_my_setup.prj" - "C:\Program Files\CPFD\Barracuda\20.0.1\bin\cpfd.x.exe" kuipers_my_s...

Barracuda release  20.0.1
Solver version     20.0.1.x023
Build date         2020-07-02 21:45:47 UTC
Restart IC version 2006


Files listed in TecplotOutputList.log exist. (R)emove, (O)verwrite, (D)isplay, (H)elp or (Q)uit? r

Output files listed in list_outputfiles.log exist. (R)emove, (A)ppend, (D)isplay, (H)elp or (Q)uit? r

Particle randomization done.

Particle randomization done.

Particle initialization adjustment ********

Identifying a license server with required licenses: barracuda-15
Connecting to license server "myRLMserver" at 27015@myRLMserver
Server contains the following product licenses: arena-7, arena-7-setup, arena-openmp, barracuda-15, barracuda-15-setup,
 barracuda-15-chem, barracuda-15-setup-chem, barracuda-gpu

Licenses found.  Proceeding to checkout licenses from identified server
Connecting to license server "myRLMserver" at 27015@myRLMserver
Checking out barracuda-15 (1)... license checked out
License(s) checked out from server: myRLMserver (27015@myRLMserver)

Reprise Project tag: path:C:\barracuda_training\2_Kuipers\my_setup/kuipers_my_setup.prj

-----------------------------------------------------------------------------------------

          t          dt  Vol      Vol   u         u   v         v   w         w    p         p
          s           s  itr      err itr       err itr       err itr       err  itr       err   CFL Low Med  Hi  R
 ------------ --------- --- --------- --- --------- --- --------- --- --------- ---- --------- ----- --- --- --- --
 0.00000e+00  1.000e-04 000  0.00e+00 000  0.00e+00 000  0.00e+00 000  0.00e+00    0  0.00e+00  0.00 344   0   0  0
Writing bvr.00000.plt files
 1.00000e-04  1.000e-04 002  2.12e-08 001  0.00e+00 000  0.00e+00 002  1.13e-12   26  9.44e-07  0.04 343   0   0  0
 2.00000e-04  1.000e-04 002  4.57e-08 003  1.73e-09 000  0.00e+00 003  3.28e-09   21  7.31e-07  0.04 343   0   0  0
 3.00000e-04  1.000e-04 002  6.97e-08 003  1.33e-09 000  0.00e+00 003  2.09e-09   19  6.17e-07  0.04 341   0   0  0
```

If CFL number is significantly below 1, increase the time step!

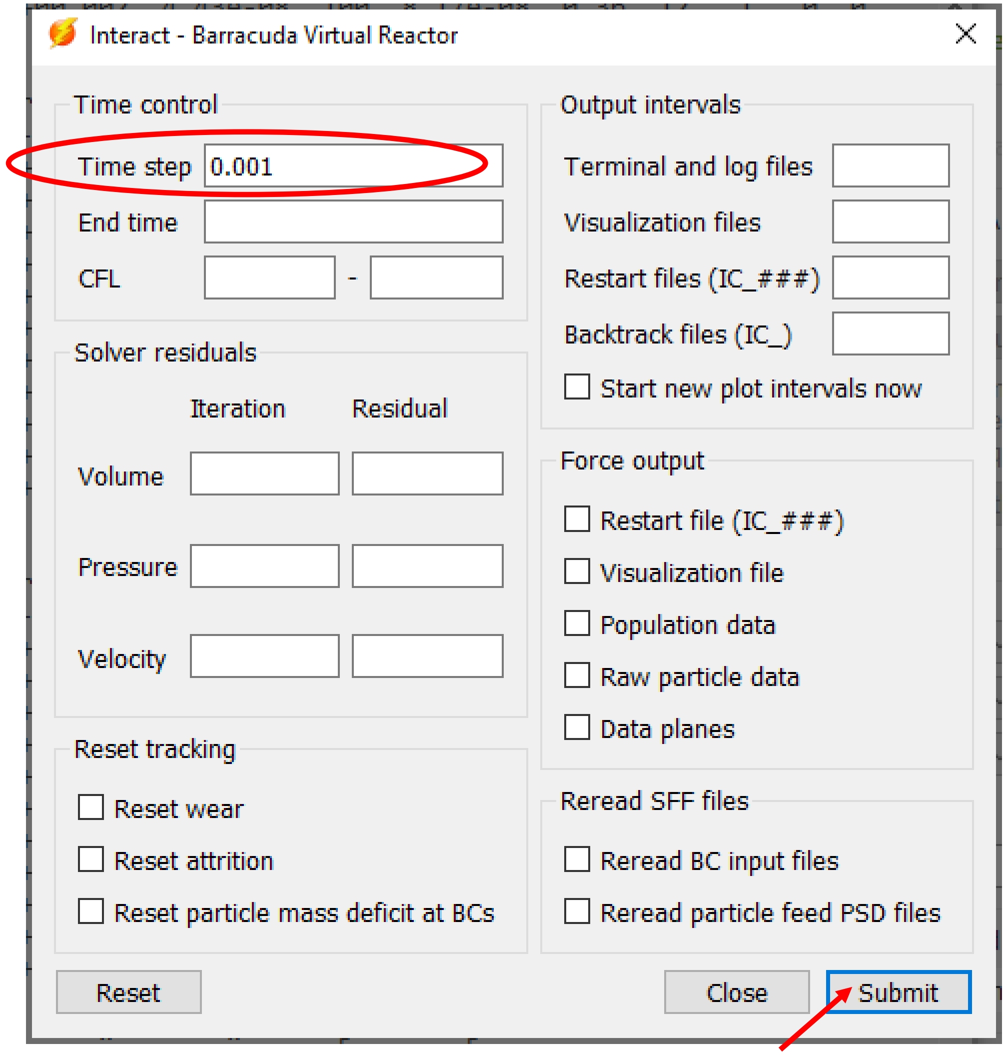
Dynamic Solver Interaction

It is possible to dynamically interact with the Barracuda solver while the calculation is running.

Click on Interact  **or** 

Only enter information you wish to change:

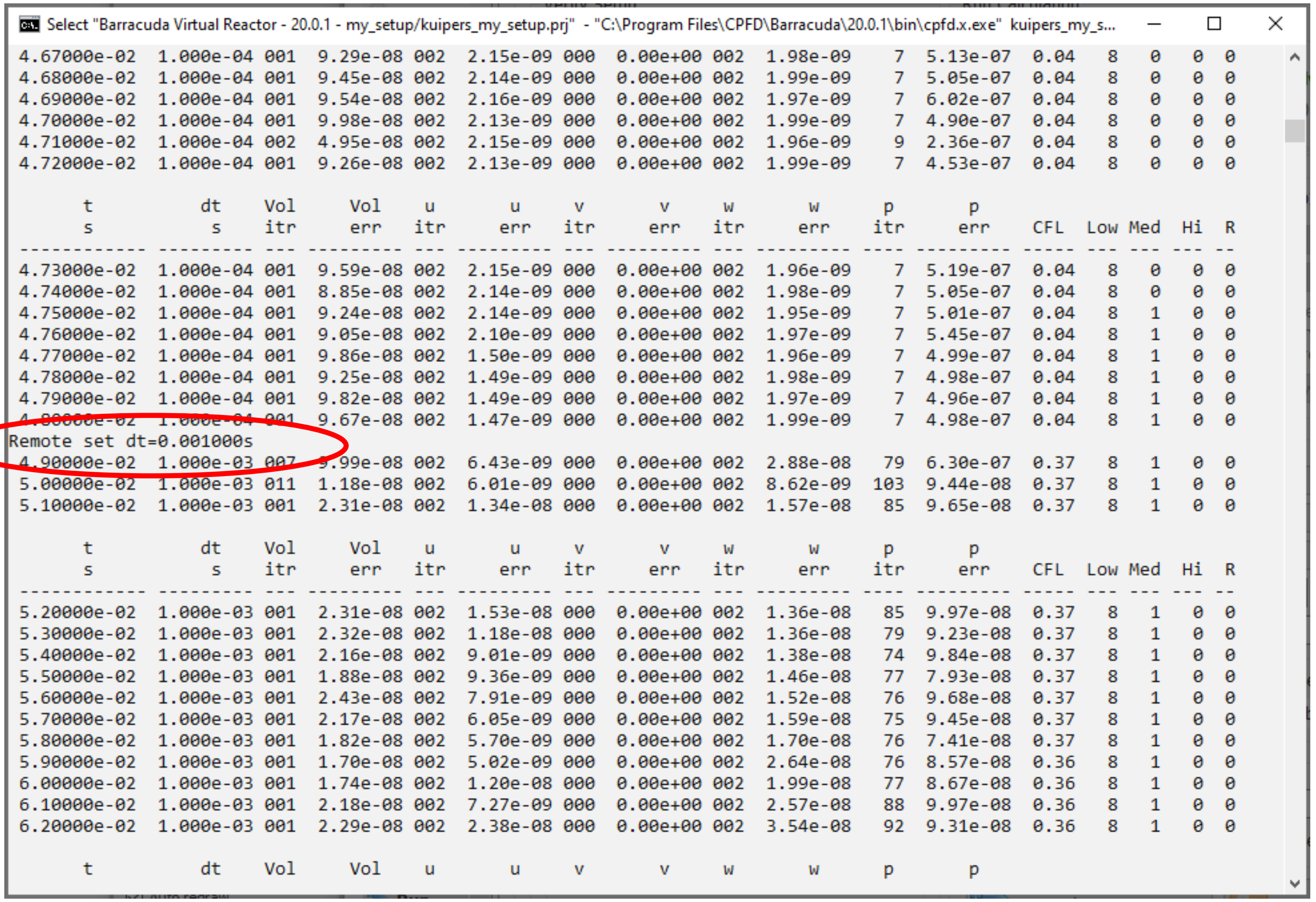
- Enter Time step of 0.001 s
- Click Submit



Dynamic Solver Interaction

Notice the solver begins running at the new time step

Tip: It is generally recommended to slowly raise the time step.



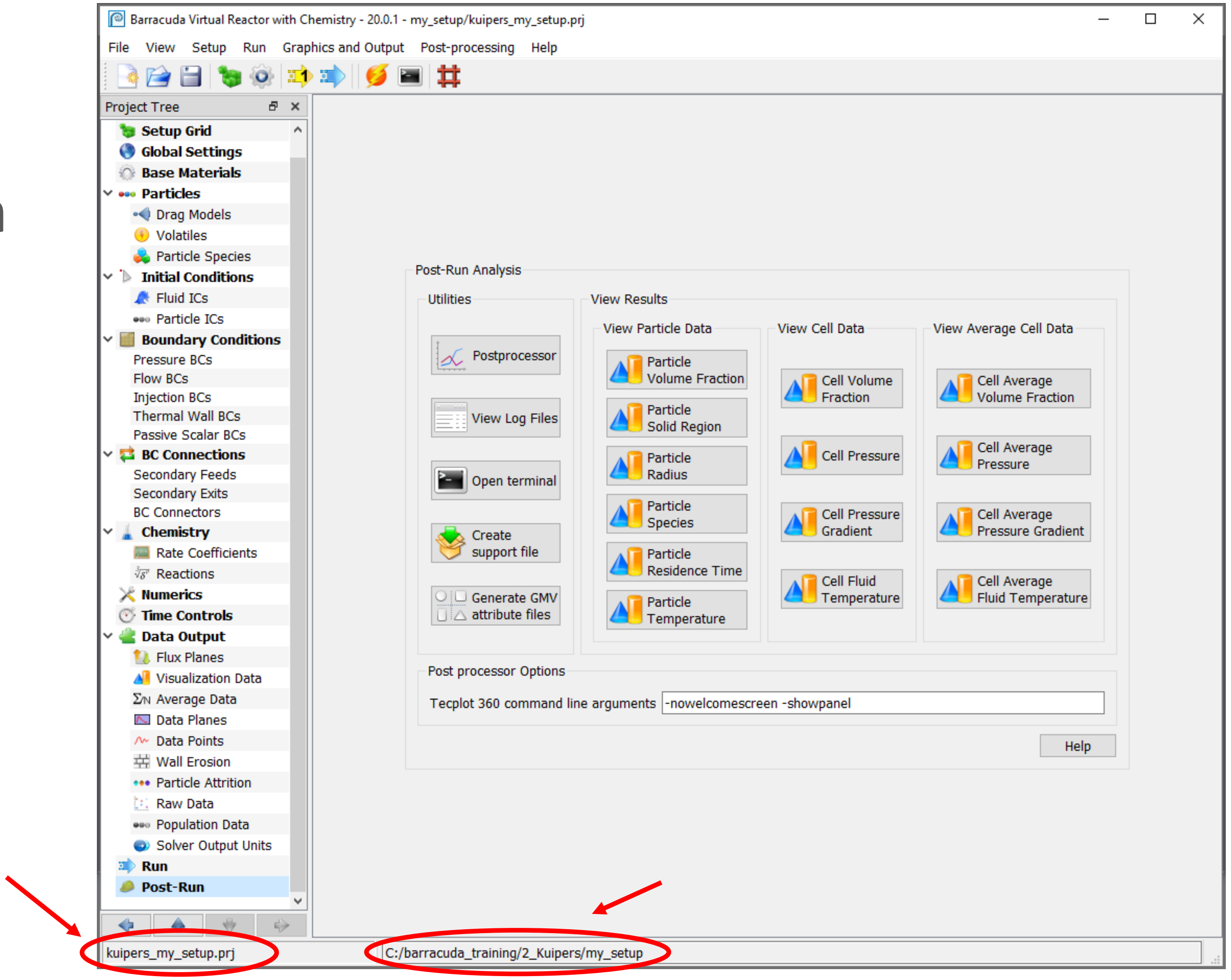
```
Select "Barracuda Virtual Reactor - 20.0.1 - my_setup/kuipers_my_setup.prj" - "C:\Program Files\CPFD\Barracuda\20.0.1\bin\cpfd.x.exe" kuipers_my_s...
4.67000e-02 1.000e-04 001 9.29e-08 002 2.15e-09 000 0.00e+00 002 1.98e-09    7 5.13e-07 0.04   8   0   0  0
4.68000e-02 1.000e-04 001 9.45e-08 002 2.14e-09 000 0.00e+00 002 1.99e-09    7 5.05e-07 0.04   8   0   0  0
4.69000e-02 1.000e-04 001 9.54e-08 002 2.16e-09 000 0.00e+00 002 1.97e-09    7 6.02e-07 0.04   8   0   0  0
4.70000e-02 1.000e-04 001 9.98e-08 002 2.13e-09 000 0.00e+00 002 1.99e-09    7 4.90e-07 0.04   8   0   0  0
4.71000e-02 1.000e-04 002 4.95e-08 002 2.15e-09 000 0.00e+00 002 1.96e-09    9 2.36e-07 0.04   8   0   0  0
4.72000e-02 1.000e-04 001 9.26e-08 002 2.13e-09 000 0.00e+00 002 1.99e-09    7 4.53e-07 0.04   8   0   0  0

      t            dt     Vol     Vol    u        u    v        v    w        w    p        p
      s             s     itr     err  itr      err  itr      err  itr      err  itr      err   CFL  Low Med  Hi  R
------------ --------- --- --------- --- --------- --- --------- --- --------- ---- --------- ----- --- --- --- --
4.73000e-02 1.000e-04 001 9.59e-08 002 2.15e-09 000 0.00e+00 002 1.96e-09    7 5.19e-07 0.04   8   0   0  0
4.74000e-02 1.000e-04 001 8.85e-08 002 2.14e-09 000 0.00e+00 002 1.98e-09    7 5.05e-07 0.04   8   0   0  0
4.75000e-02 1.000e-04 001 9.24e-08 002 2.14e-09 000 0.00e+00 002 1.95e-09    7 5.01e-07 0.04   8   1   0  0
4.76000e-02 1.000e-04 001 9.05e-08 002 2.10e-09 000 0.00e+00 002 1.97e-09    7 5.45e-07 0.04   8   1   0  0
4.77000e-02 1.000e-04 001 9.86e-08 002 1.50e-09 000 0.00e+00 002 1.96e-09    7 4.99e-07 0.04   8   1   0  0
4.78000e-02 1.000e-04 001 9.25e-08 002 1.49e-09 000 0.00e+00 002 1.98e-09    7 4.98e-07 0.04   8   1   0  0
4.79000e-02 1.000e-04 001 9.82e-08 002 1.49e-09 000 0.00e+00 002 1.97e-09    7 4.96e-07 0.04   8   1   0  0
4.80000e-02 1.000e-04 001 9.67e-08 002 1.47e-09 000 0.00e+00 002 1.99e-09    7 4.98e-07 0.04   8   1   0  0
Remote set dt=0.001000s
4.90000e-02 1.000e-03 007 9.99e-08 002 6.43e-09 000 0.00e+00 002 2.88e-08   79 6.30e-07 0.37   8   1   0  0
5.00000e-02 1.000e-03 011 1.18e-08 002 6.01e-09 000 0.00e+00 002 8.62e-09  103 9.44e-08 0.37   8   1   0  0
5.10000e-02 1.000e-03 001 2.31e-08 002 1.34e-08 000 0.00e+00 002 1.57e-08   85 9.65e-08 0.37   8   1   0  0

      t            dt     Vol     Vol    u        u    v        v    w        w    p        p
      s             s     itr     err  itr      err  itr      err  itr      err  itr      err   CFL  Low Med  Hi  R
------------ --------- --- --------- --- --------- --- --------- --- --------- ---- --------- ----- --- --- --- --
5.20000e-02 1.000e-03 001 2.31e-08 002 1.53e-08 000 0.00e+00 002 1.36e-08   85 9.97e-08 0.37   8   1   0  0
5.30000e-02 1.000e-03 001 2.32e-08 002 1.18e-08 000 0.00e+00 002 1.36e-08   79 9.23e-08 0.37   8   1   0  0
5.40000e-02 1.000e-03 001 2.16e-08 002 9.01e-09 000 0.00e+00 002 1.38e-08   74 9.84e-08 0.37   8   1   0  0
5.50000e-02 1.000e-03 001 1.88e-08 002 9.36e-09 000 0.00e+00 002 1.46e-08   77 7.93e-08 0.37   8   1   0  0
5.60000e-02 1.000e-03 001 2.43e-08 002 7.91e-09 000 0.00e+00 002 1.52e-08   76 9.68e-08 0.37   8   1   0  0
5.70000e-02 1.000e-03 001 2.17e-08 002 6.05e-09 000 0.00e+00 002 1.59e-08   75 9.45e-08 0.37   8   1   0  0
5.80000e-02 1.000e-03 001 1.82e-08 002 5.70e-09 000 0.00e+00 002 1.70e-08   76 7.41e-08 0.37   8   1   0  0
5.90000e-02 1.000e-03 001 1.70e-08 002 5.02e-09 000 0.00e+00 002 2.64e-08   76 8.57e-08 0.36   8   1   0  0
6.00000e-02 1.000e-03 001 1.74e-08 002 1.20e-08 000 0.00e+00 002 1.99e-08   77 8.67e-08 0.36   8   1   0  0
6.10000e-02 1.000e-03 001 2.18e-08 002 7.27e-09 000 0.00e+00 002 2.57e-08   88 9.97e-08 0.36   8   1   0  0
6.20000e-02 1.000e-03 001 2.29e-08 002 2.38e-08 000 0.00e+00 002 3.54e-08   92 9.31e-08 0.36   8   1   0  0

      t            dt     Vol     Vol    u        u    v        v    w        w    p        p
```

Post Processing

Post-Run

During and after a Barracuda simulation, examine your results by clicking on the Post Run tab. You can view Tecplot files while the solver is operating

If you have reopened the Barracuda GUI, check that the project file and working directory are correct.

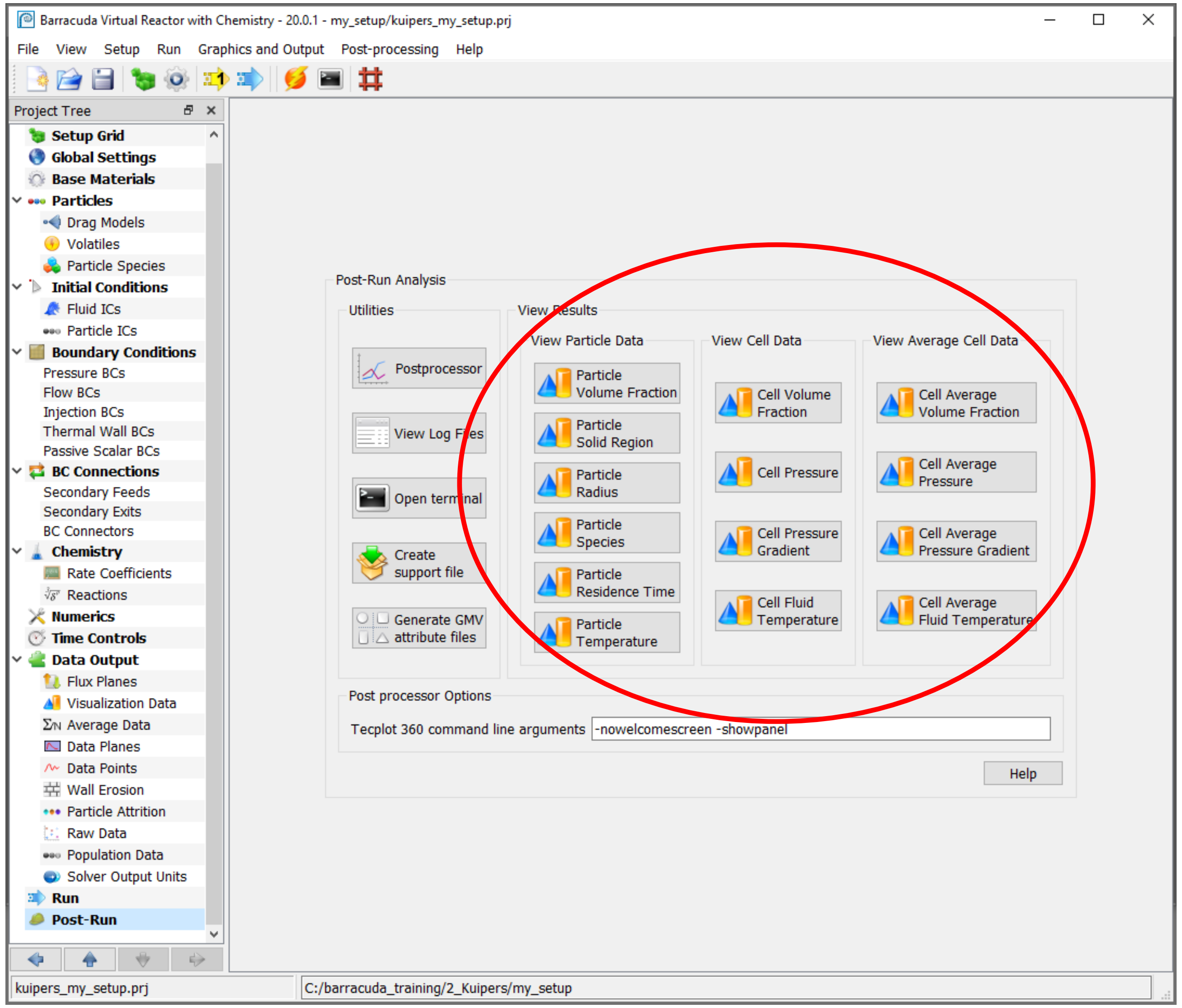


Tecplot Shortcut Buttons

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)

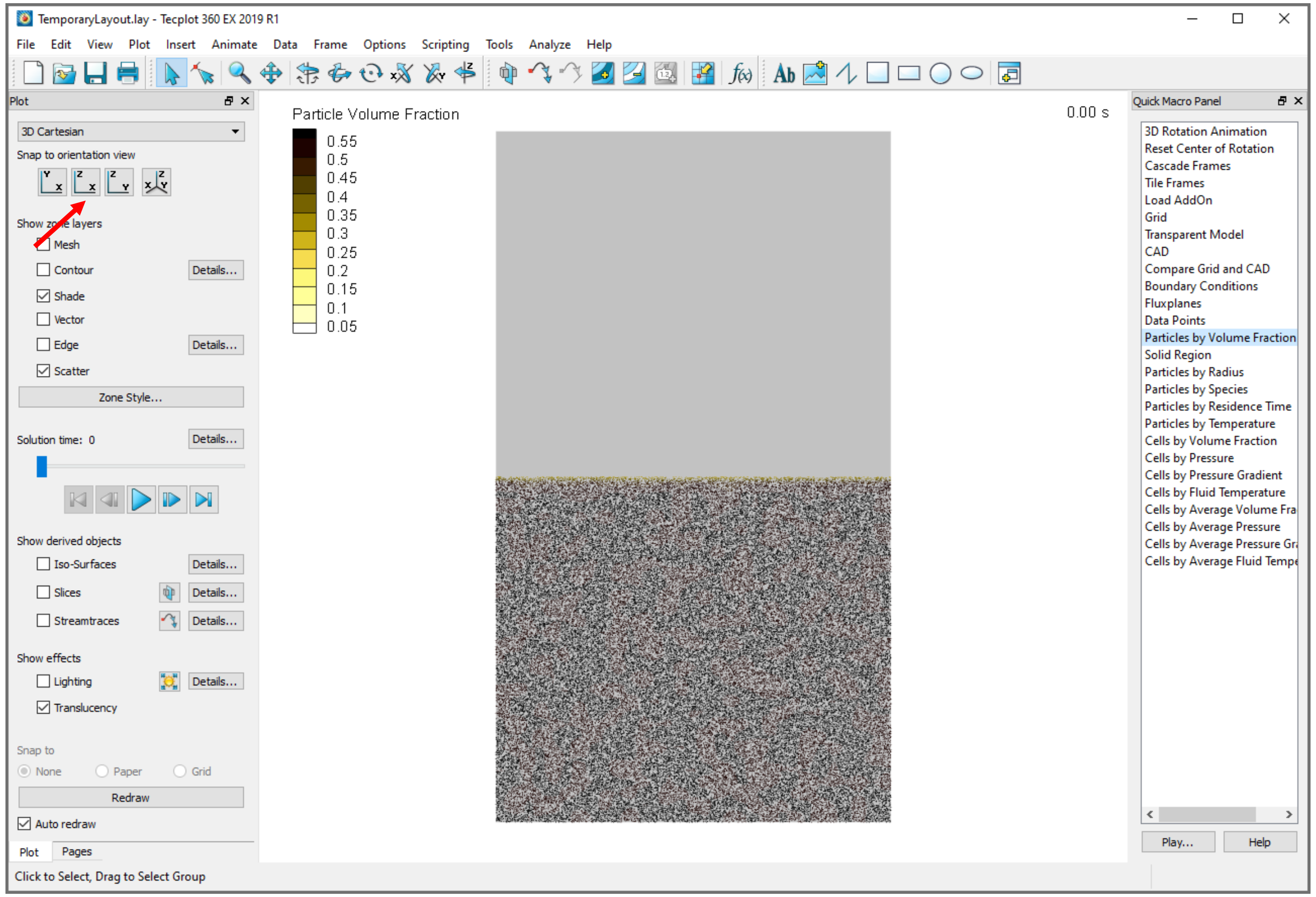


Particle Volume Fraction View

Open this view by clicking on Particle Volume Fraction in the Post-Run window

Rotate the image into view

- This can be accomplished quickly by selecting the “XZ plane” View
- For more information on manipulating the view in Tecplot, see [this video]



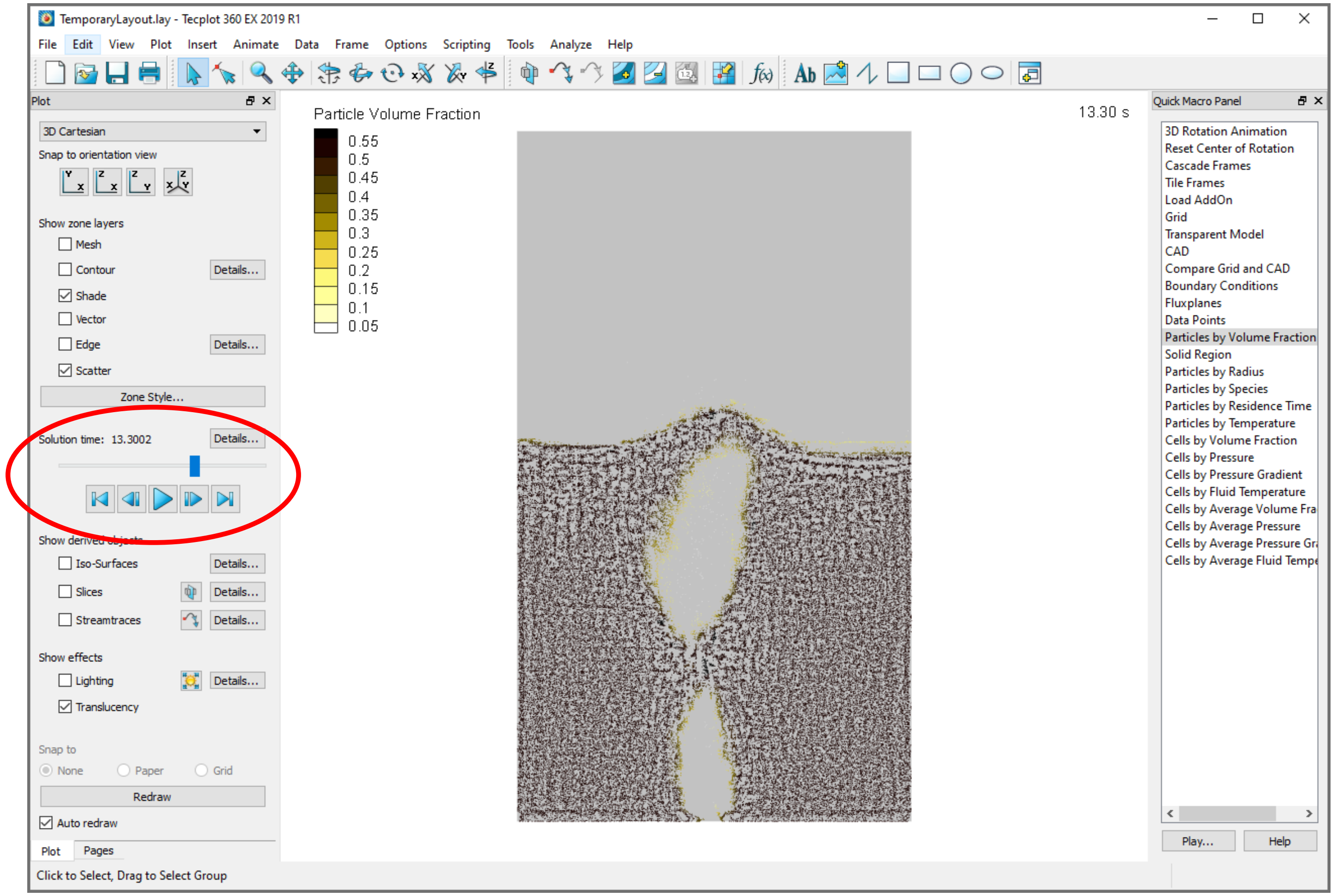
Navigating through Time

Tecplot will load data up to the time-step when you clicked on the view. If more time has elapsed and more .plt files have been created, you can scan for more files to view:

- Click on File→ Load Barracuda data…
- Click on Rescan All Data
- Click Close

To navigate through time, use the time controls located in the Plot sidebar

- Use the Play button in the center to view your simulation through time
- Use the slider to quickly navigate to a certain point in time
- The left- and right-most buttons skip to the beginning and end, and the buttons just inside them navigate to the previous or next file



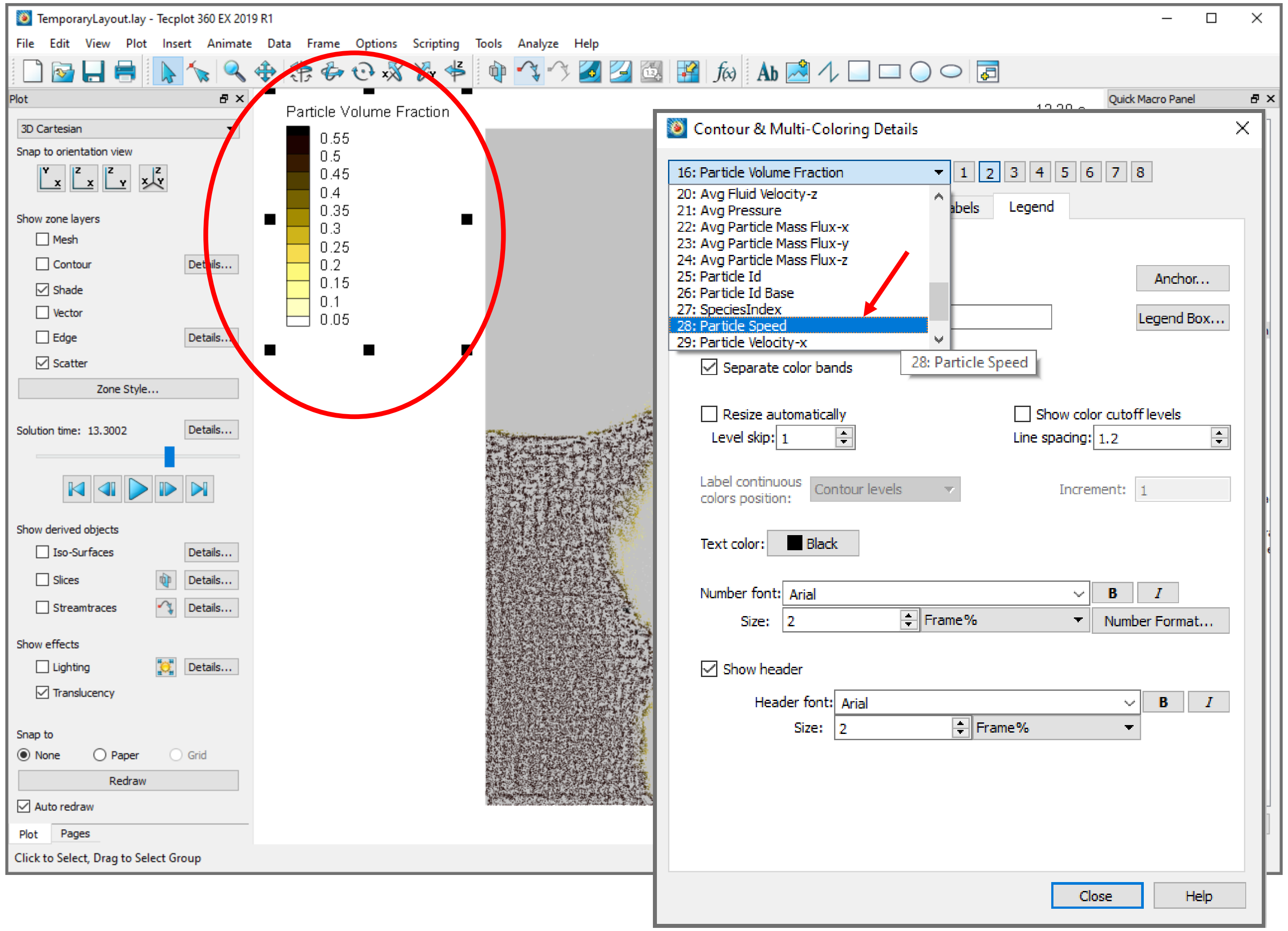
Coloring by a Different Variable

To change which data is being presented, double-click on the legend

In the drop down menu at the top left, select the data you would like to see displayed (for the next few slides, we will be using 16: Particle Volume Fraction)

For a scatter zone, you should select particle-dependent data. Likewise, for a Contour or Vector zone, you should select cell-dependent data.

Note: Only data selected for visualization data output during the setup will be available



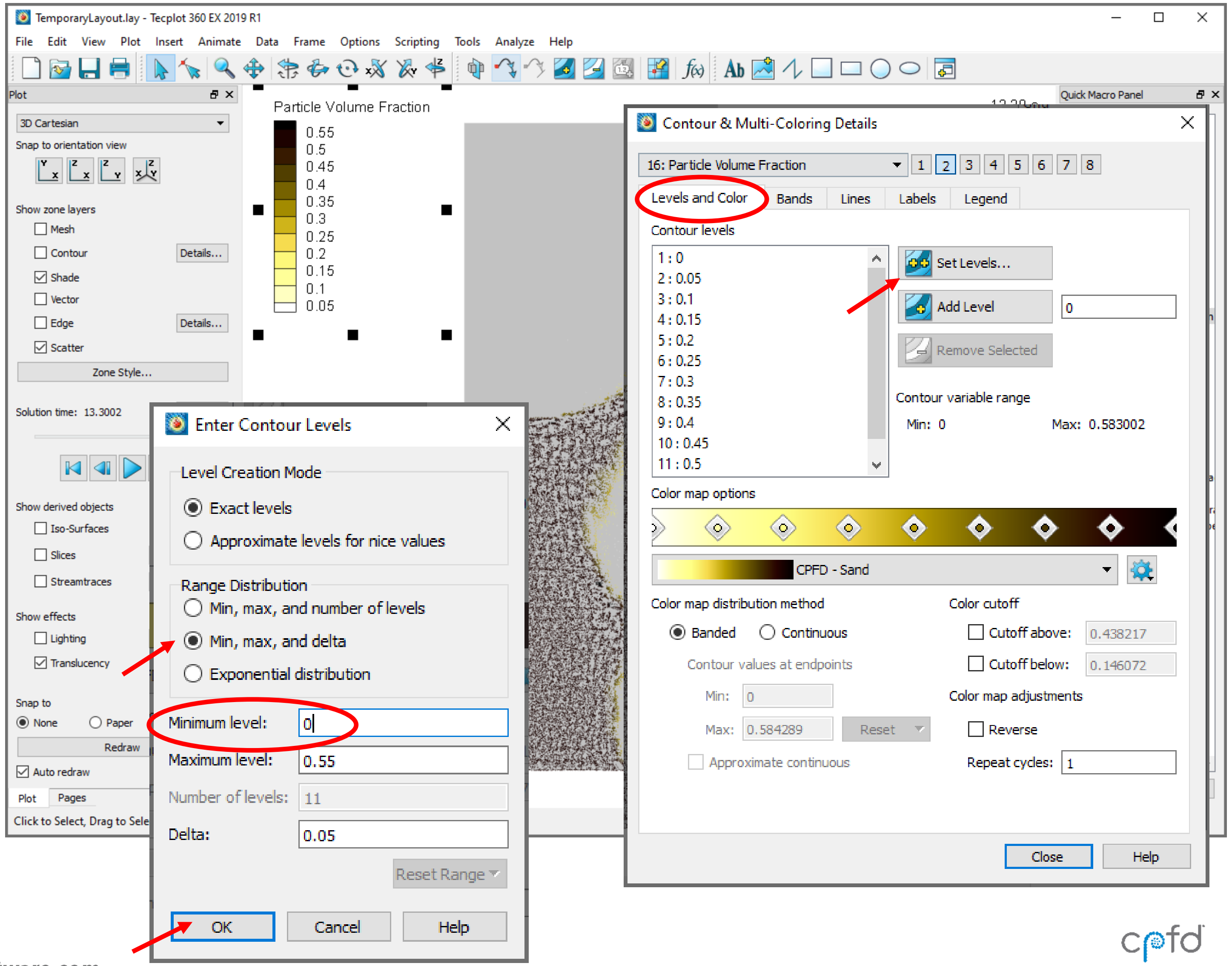
Setting Data Limits

We can control the minimum and maximum values to display

- Note: Any values above or below these respectively will be colored as if they are at this value

To change the data limits:

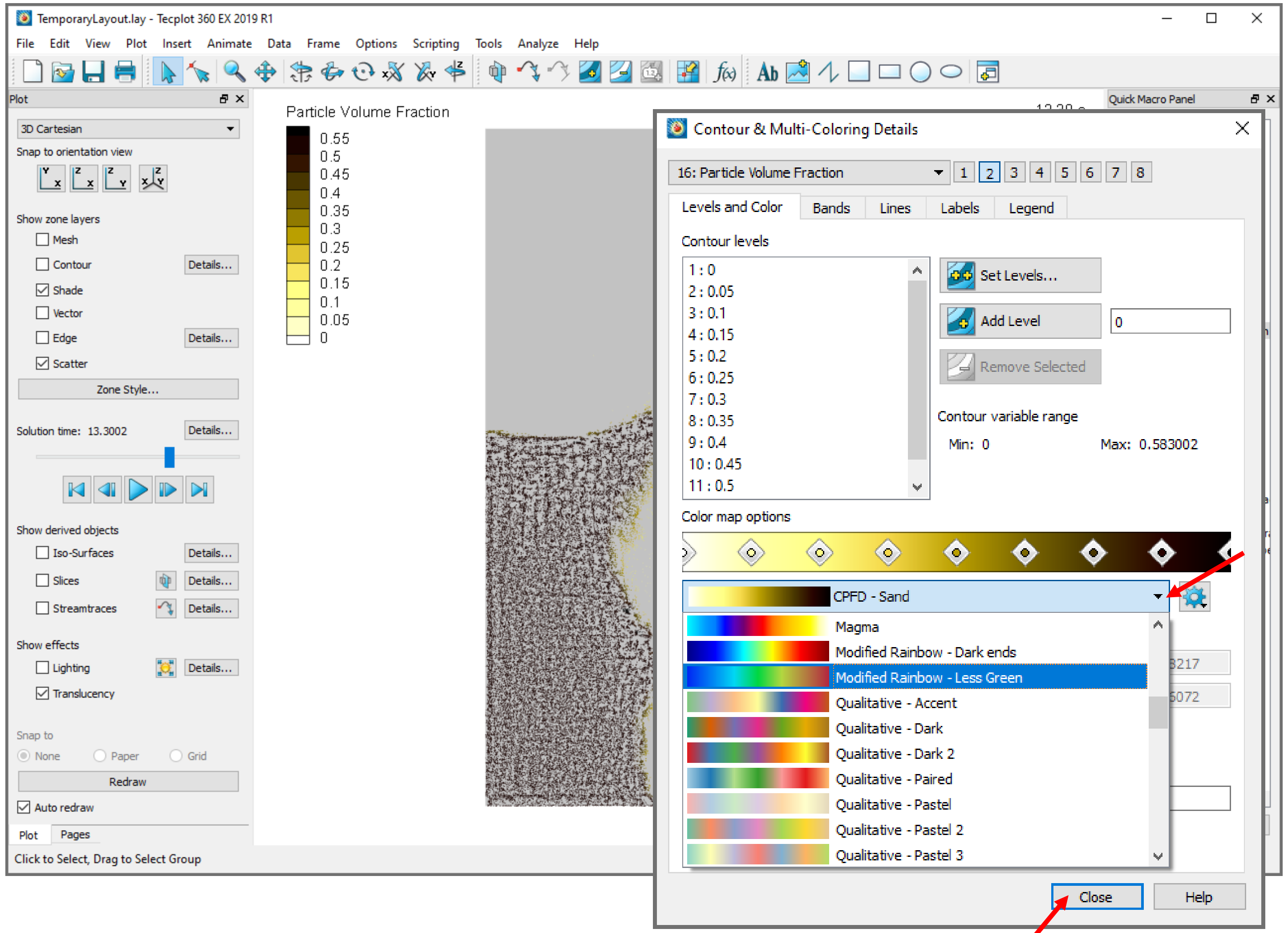
- Navigate to the Levels and Color tab
- Click on the Set Levels… button, which will bring up the Enter Contour Levels dialog
- Select Min, max, and delta for Range Distribution
- Change the Minimum level to 0
- Click OK
- Observe the minimum in the legend has changed to 0



Changing the Color Bar

To change how the view is colored:

- Click on the drop down menu in the Color map options section
- Select Modified Rainbow – Less Green
- Click on Close

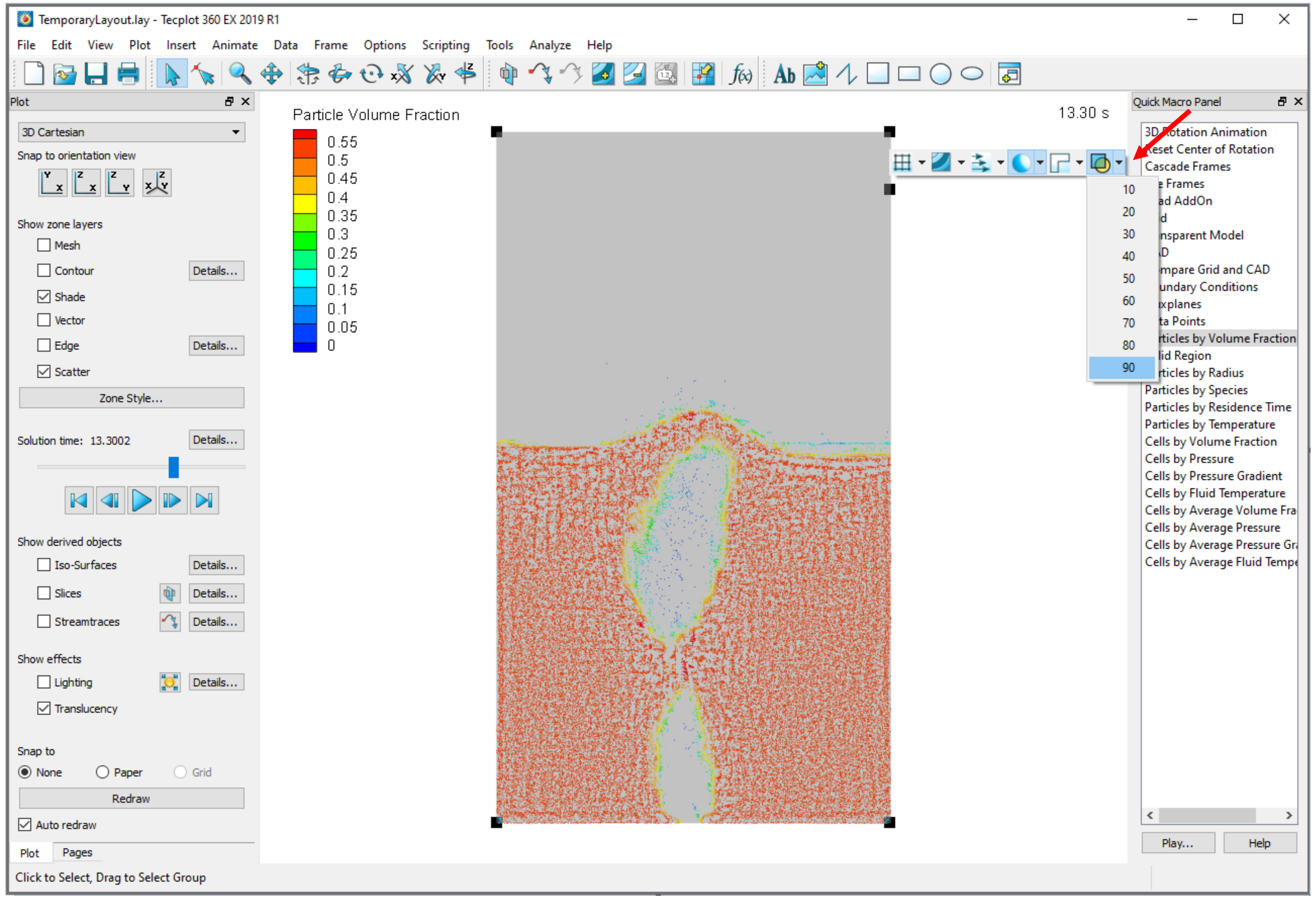


Translucent Geometry

The default view of model geometry is at 80% translucency

While this often looks good for many views, in some situations you will want to modify this value:

- Right-click the geometry
- Click the rightmost figure to toggle the translucency
- Click the dropdown menu next to it to select a translucency value
- Try out different values for the model geometry translucency and see which looks best for this project (90% is shown in the next slides)

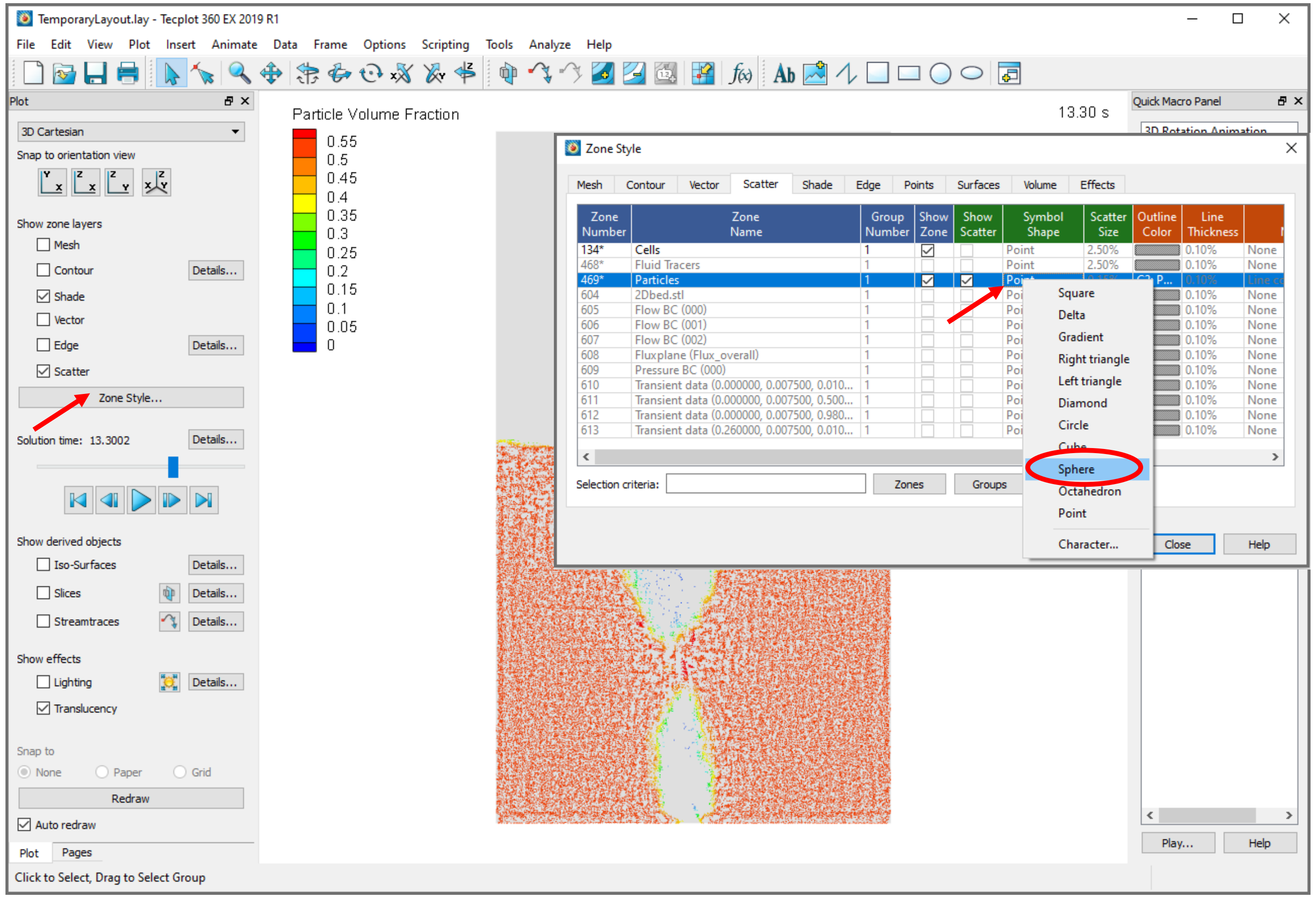


Changing Scatter Symbol Shape and Size

The default scatter symbol shape is one-pixel points, which is great for rendering a very large number of particles in a reasonable amount of time.

If you have not so many particles and want to show them a little larger:

- Click on Zone Style…
- On the Scatter tab, right-click on Point in the Symbol Shape column of the Particles row
  - Select Sphere
- Right-click on Scatter Size column of Particles row
  - Click on Enter…
  - Enter 0.3
  - Click OK

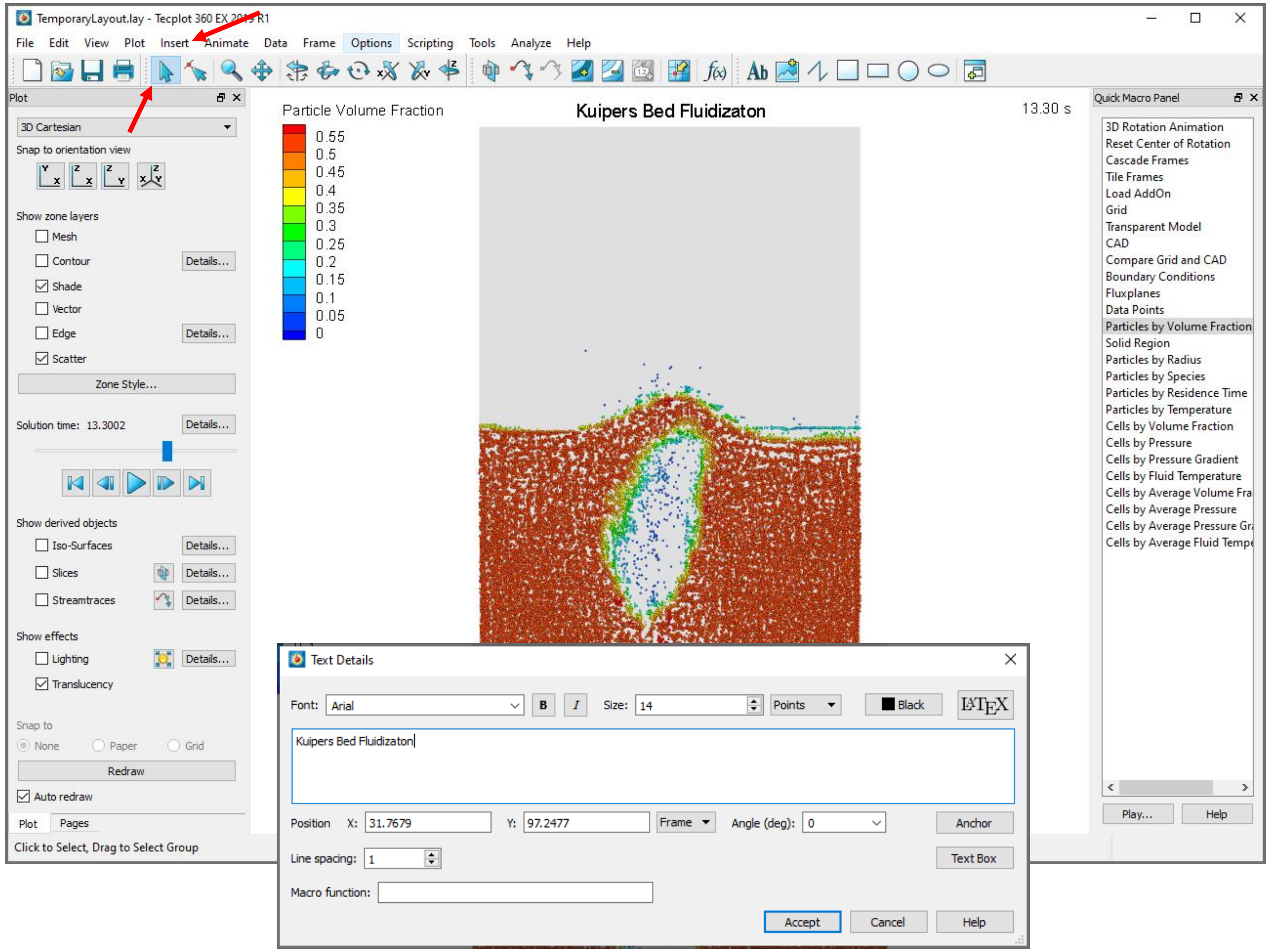


Adding a Title

Select Insert → Text and select a location in the view to raise the Text Details window

- Here you can adjust the font, size, color, location, etc of the text
- Click Accept when your text is ready

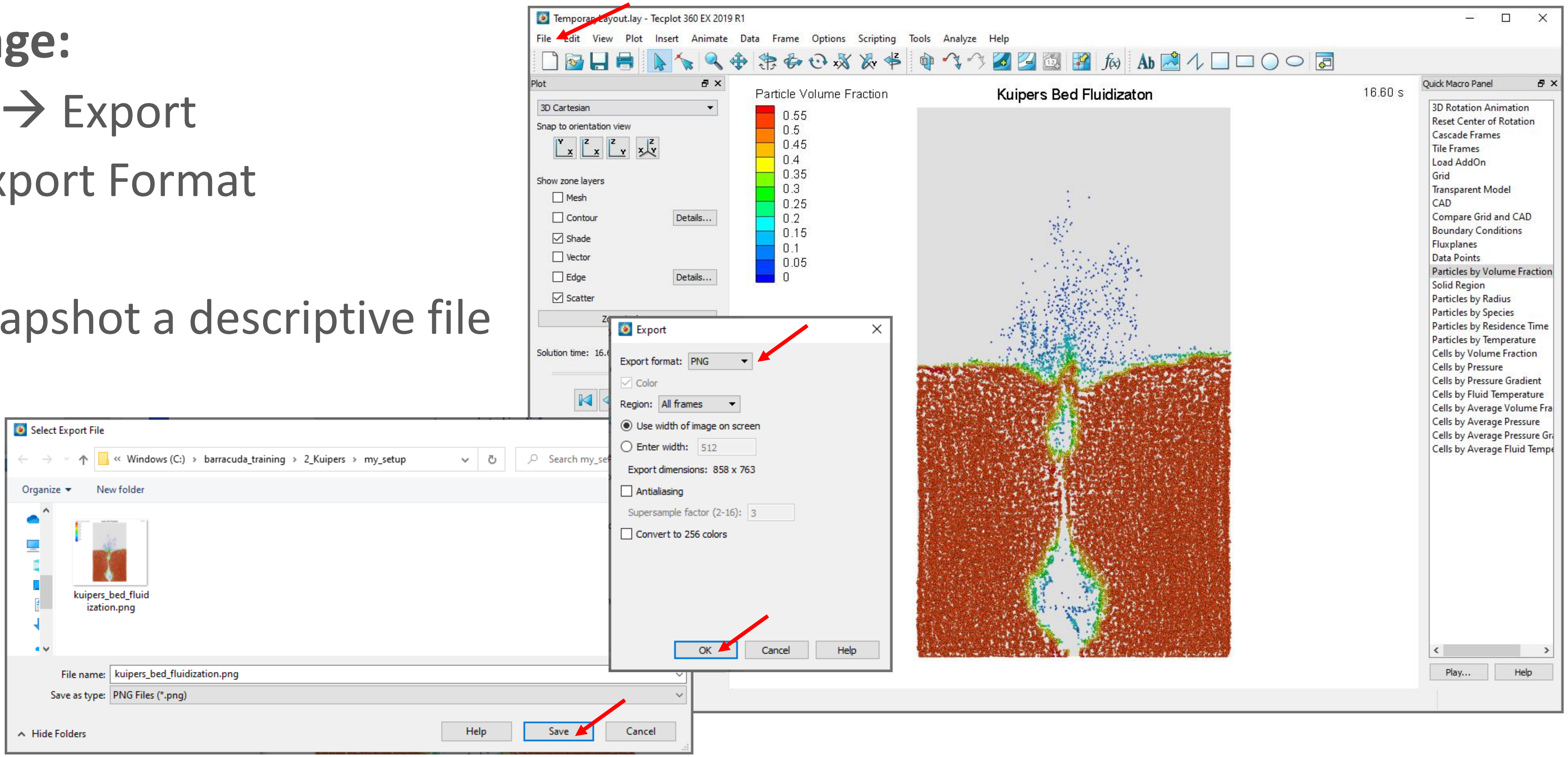
Once entered, any text can be moved around with the selection tool



Exporting an image

To create an image:

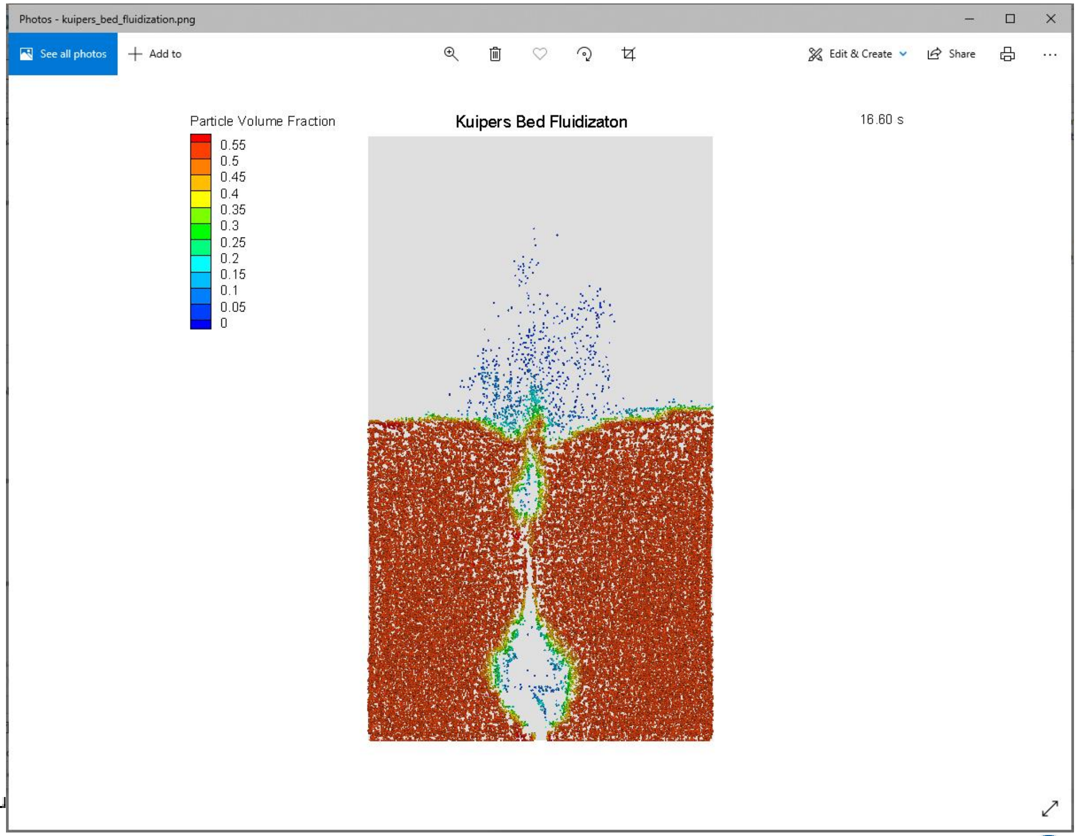
- Click on File → Export
- Select the Export Format
- Click OK
- Give your snapshot a descriptive file name
- Click Save

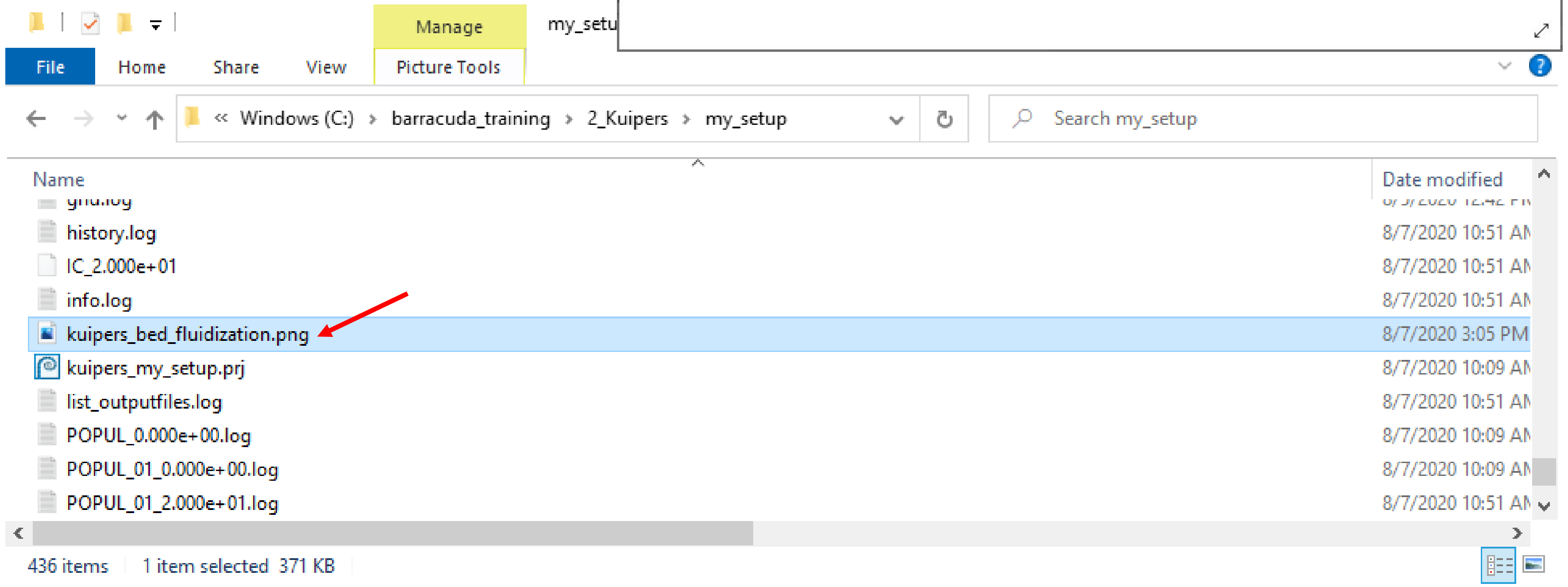


Viewing an image

To view your snapshot:

- Navigate to your project file directory
- Double-click on snapshot file name

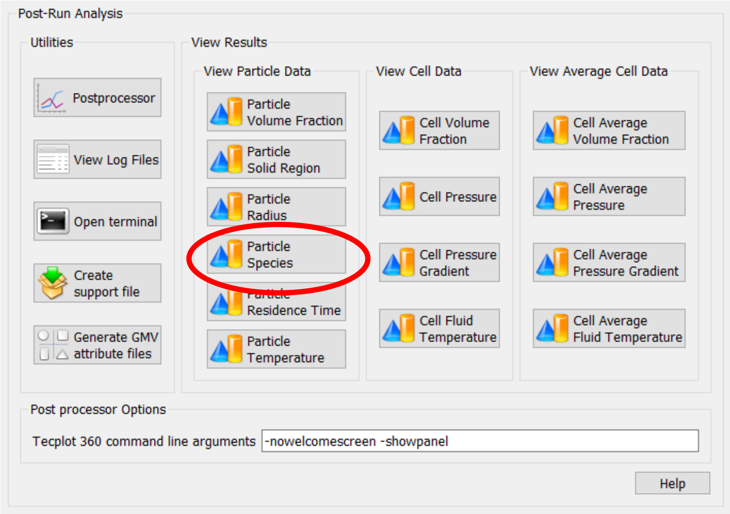


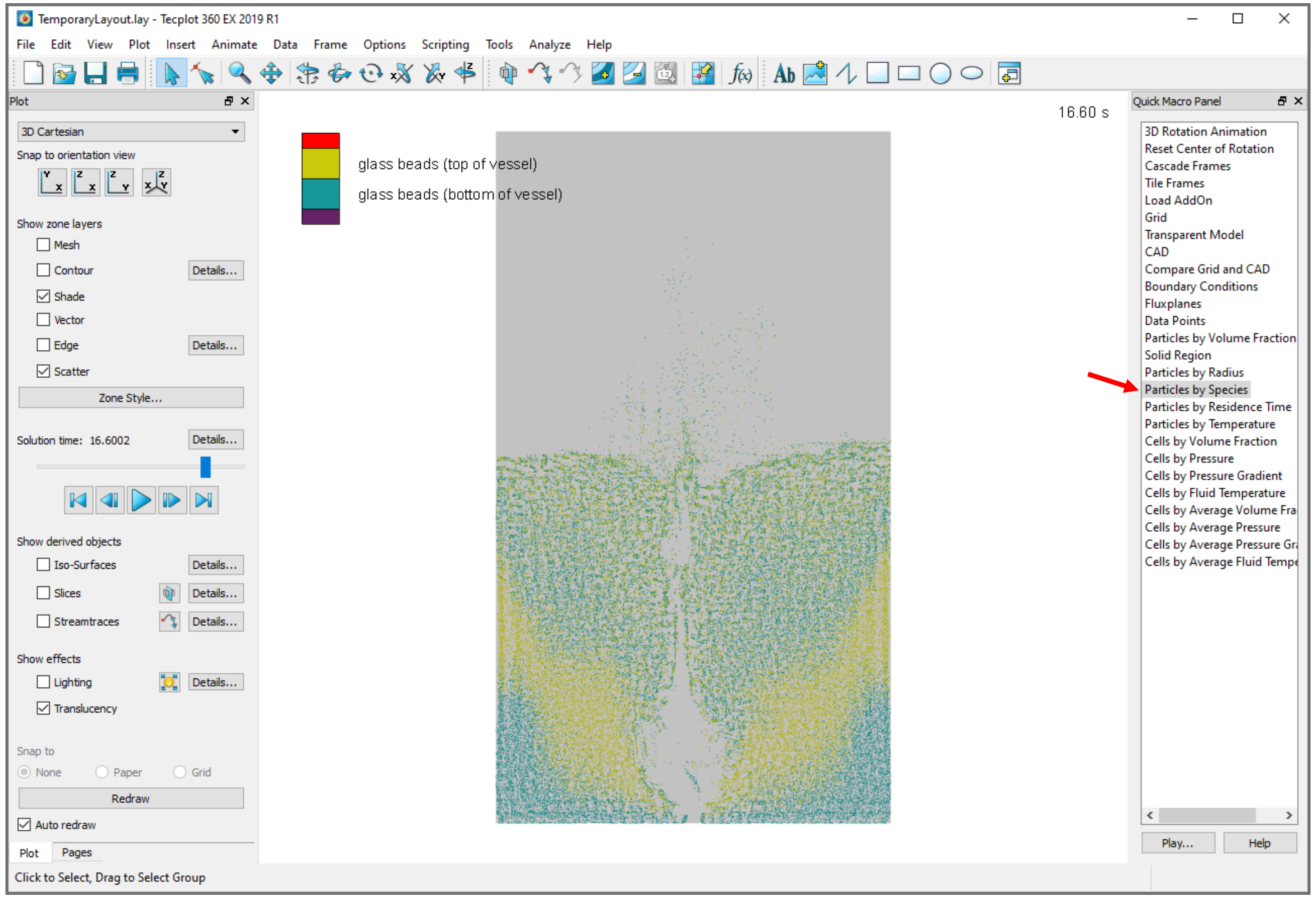


Particle Mixing in Kuipers Bed

Particle mixing can be showed best with an animation of the Kuipers bed through time.

Double-click on Particles by Species in the Quick Macro Panel or the Particle Species button in the Post-Run window.





Making an Animation

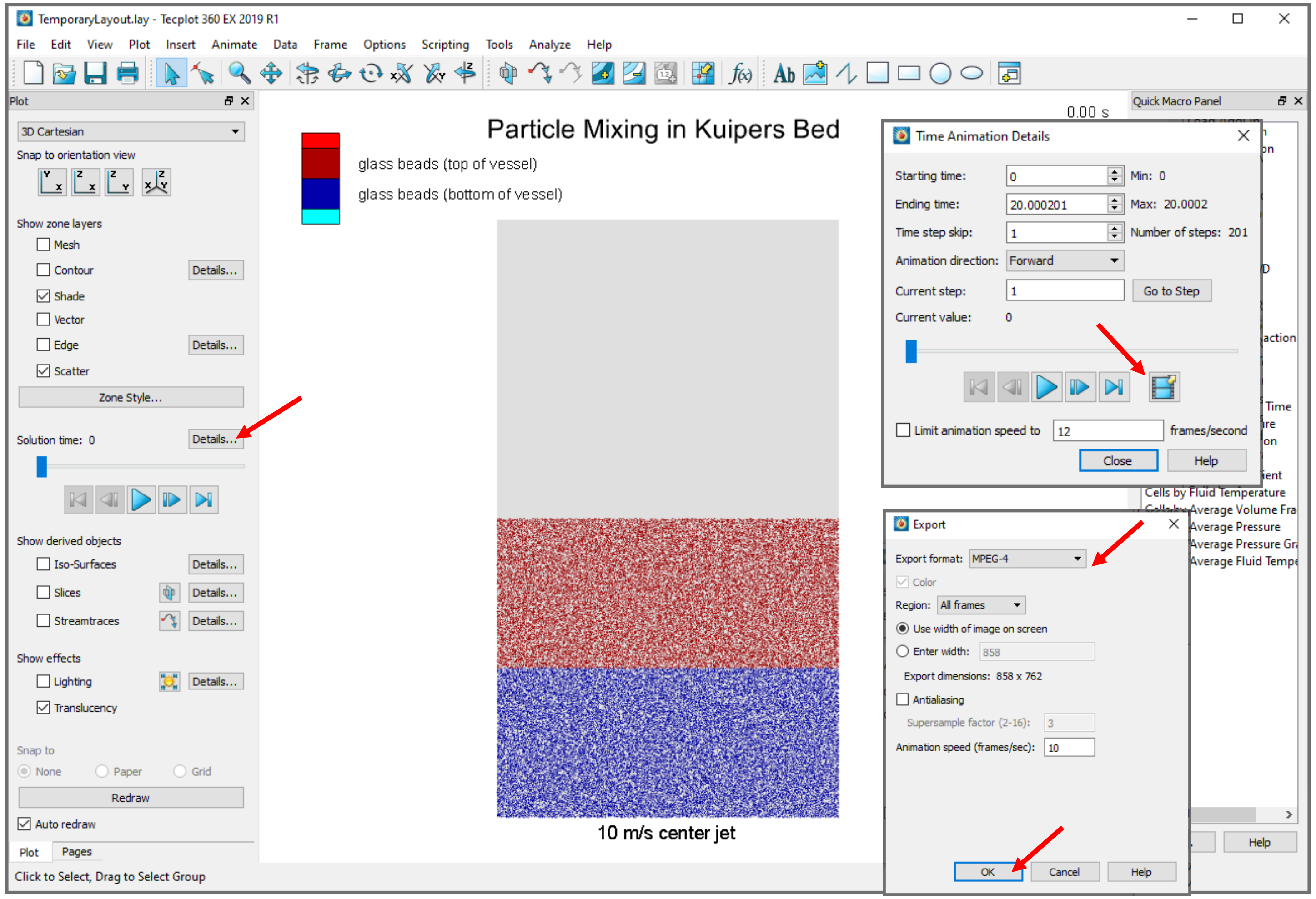
Create this view:

- Color map : Doppler
- Adjusted zoom and location of geometry
- Added top and bottom title

Create an animation:

- Click on Details… next to Solution time
- Verify the settings in the Time Animation Details dialog
- Click on the Film Reel icon
- Select the Export format
- Click OK
- Give descriptive file name
- Click Save

To view the animation, navigate to the simulation directory and double-click on animation file name



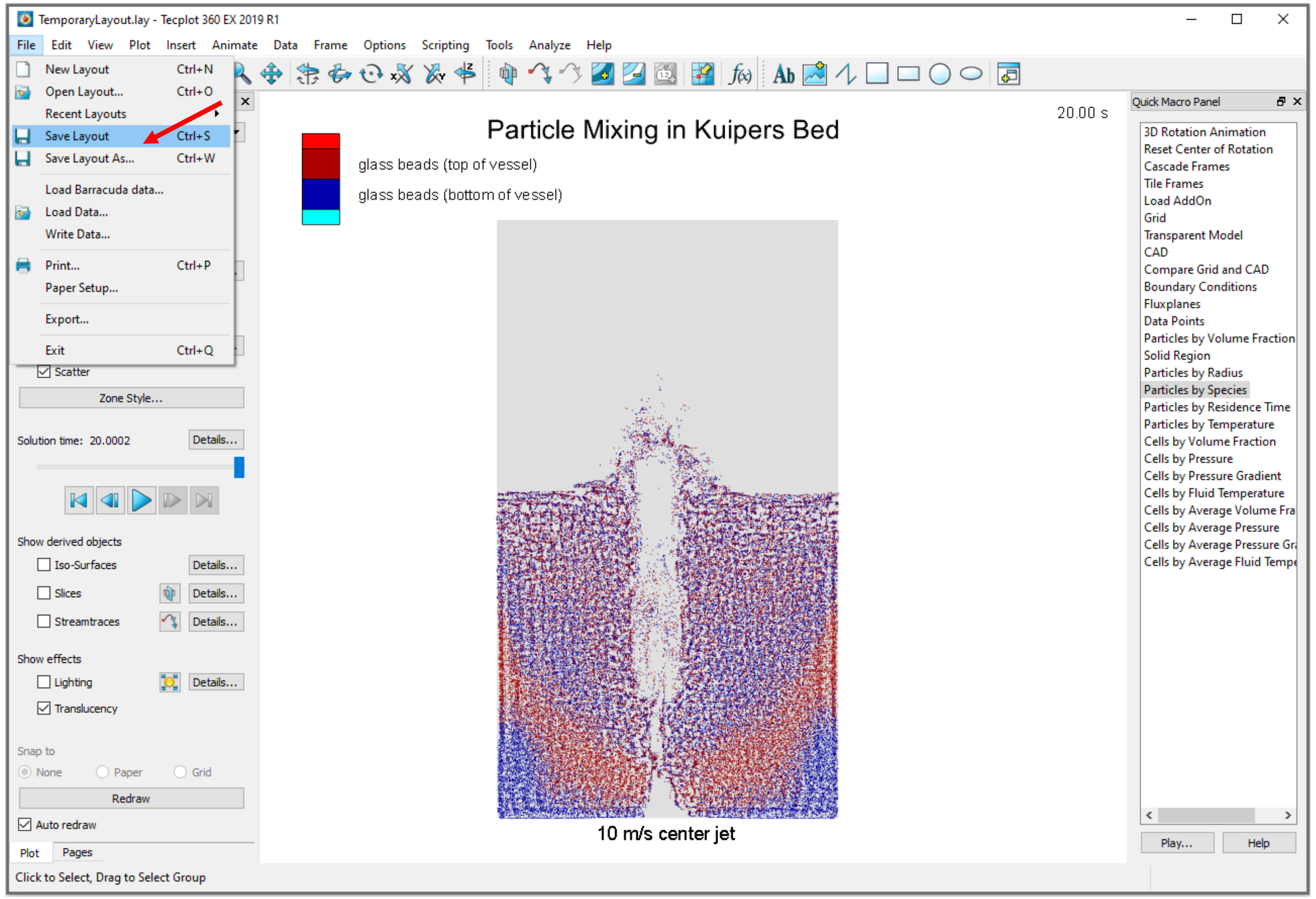
Saving an Layout File

When finished setting up a non-default data view, it's good practice to save it

Select File → Save Layout… to name and save a custom layout

A layout file will save all the requisite information to recreate the view

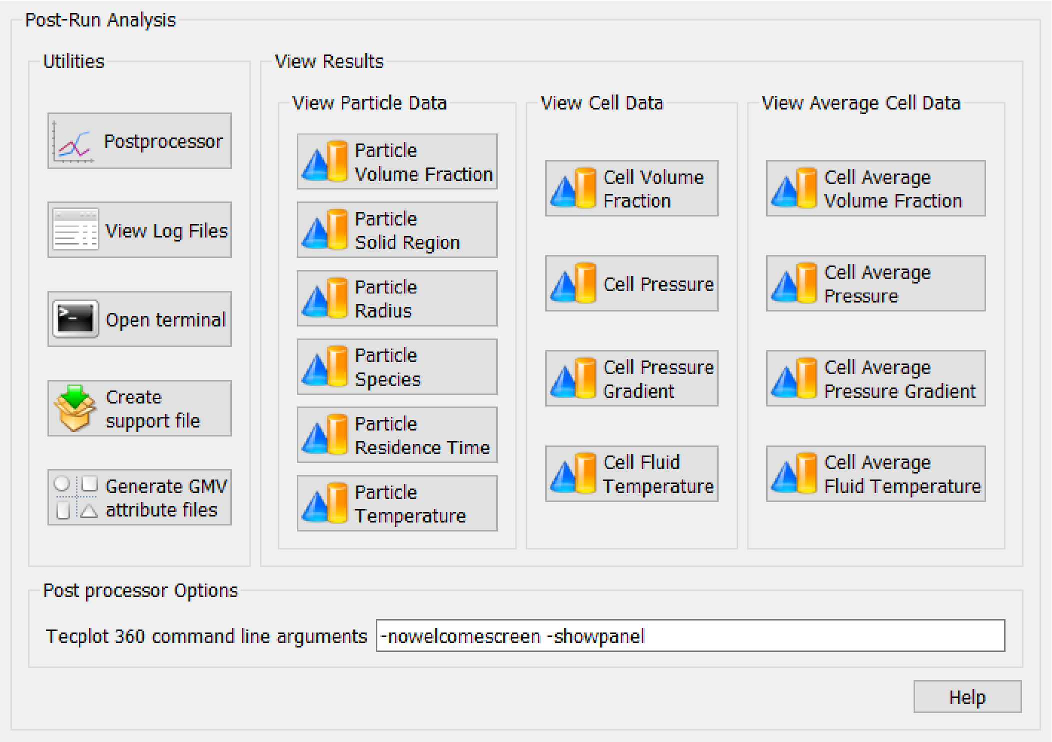
Note: Layout files can be applied to other projects with different data and different geometry. This can be helpful for quickly creating animations and snapshots in multiple directories.

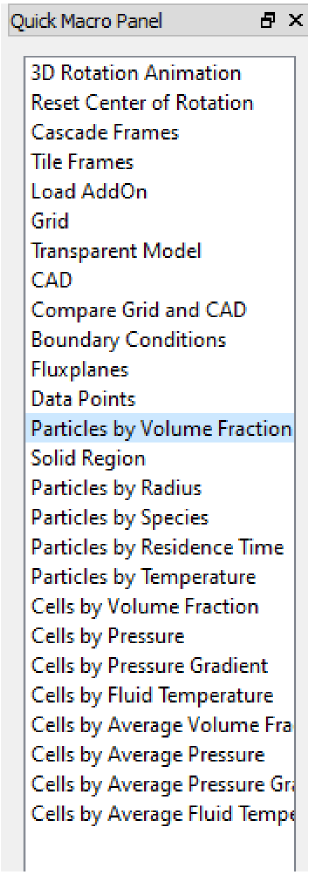


Additional Tecplot Shortcuts

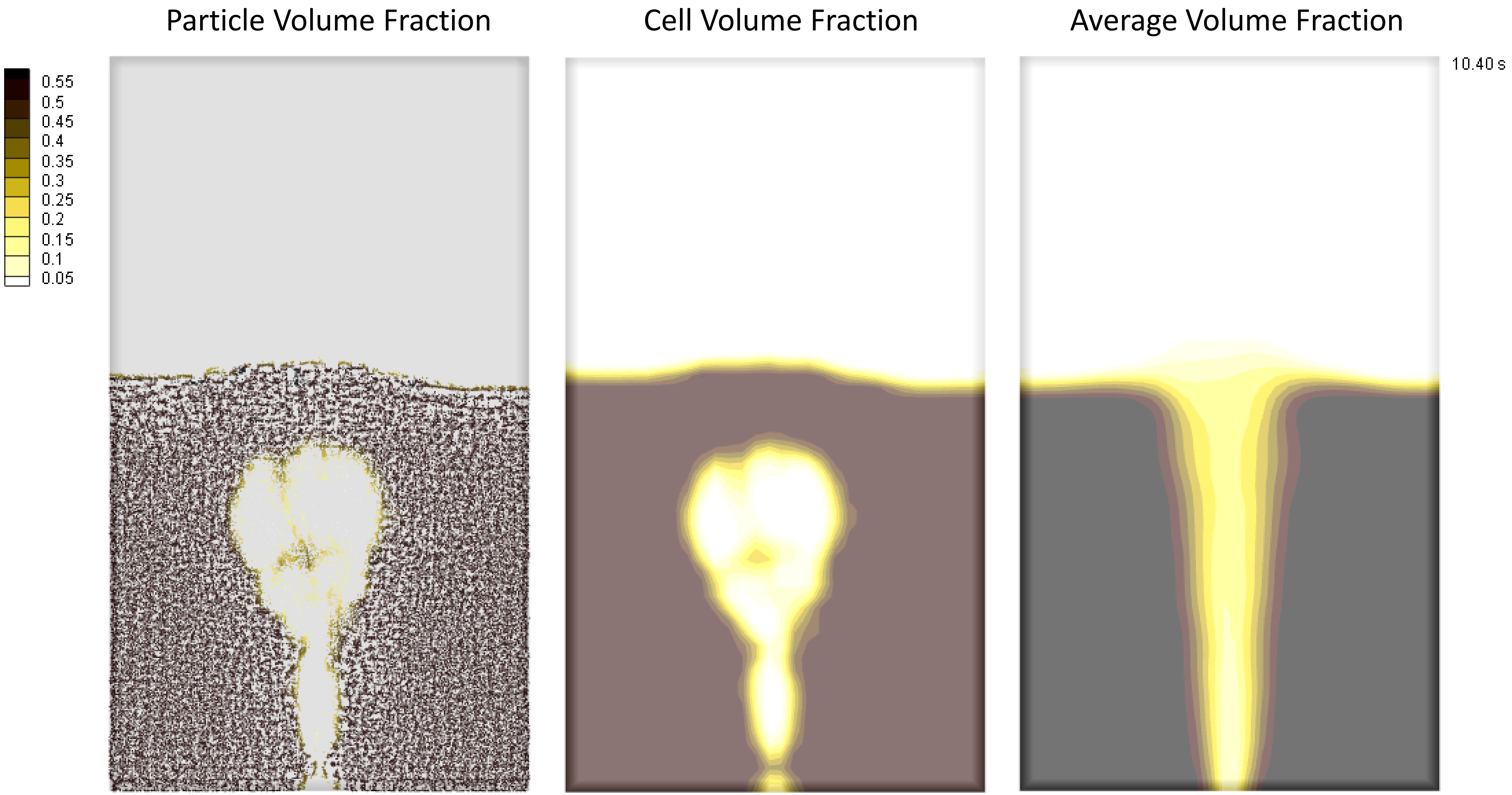
All the shortcut buttons in View Results are available in the Quick Macro Panel as well. Once in Tecplot, it is best to use the Quick Macro Panel to view results.

- 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?





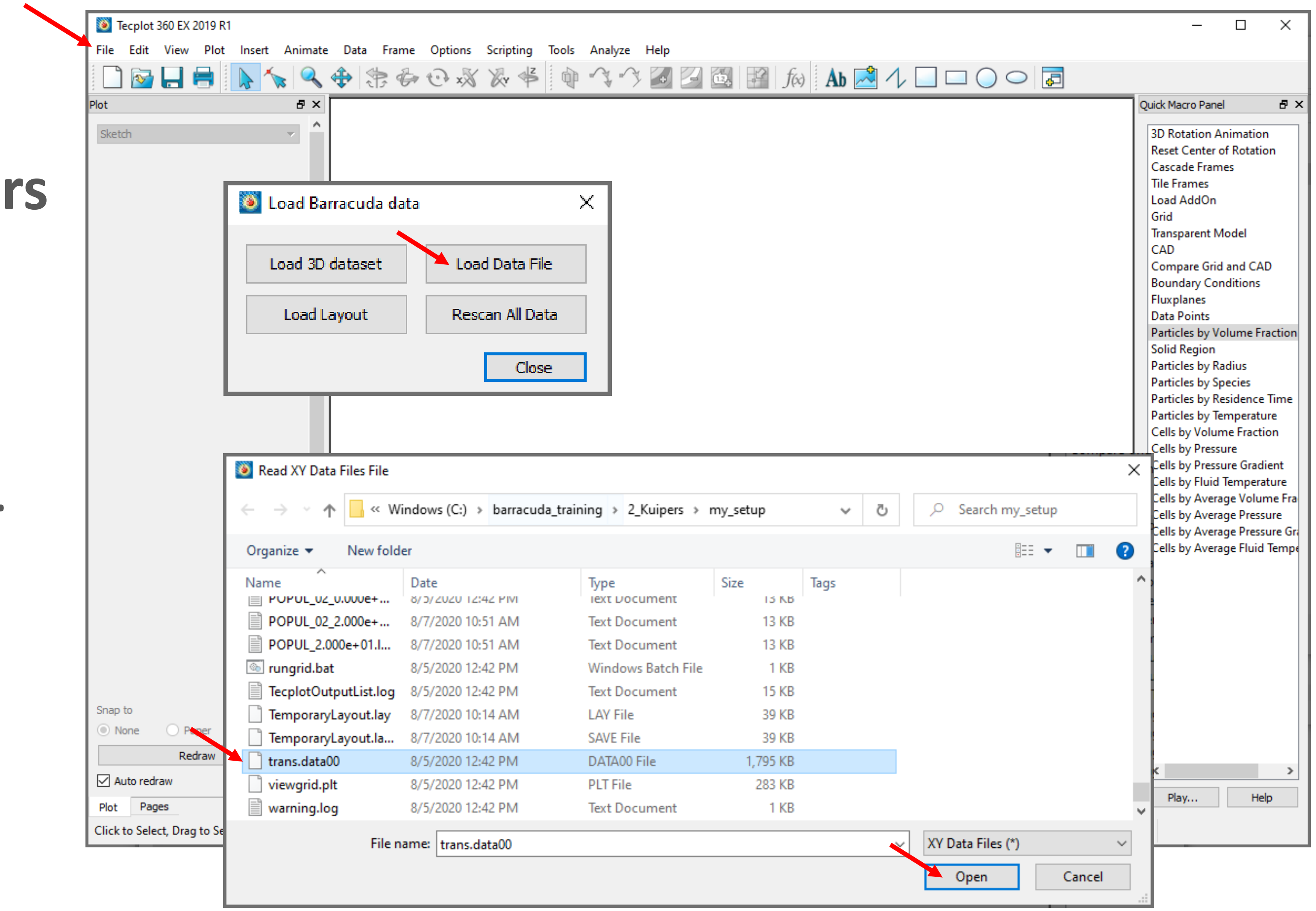
Comparing Results



Loading Barracuda Data for xy plots

When setting up the simulation, we defined Data Points to measure pressure at specific cells in the Kuipers bed. Let's plot the data from those points.

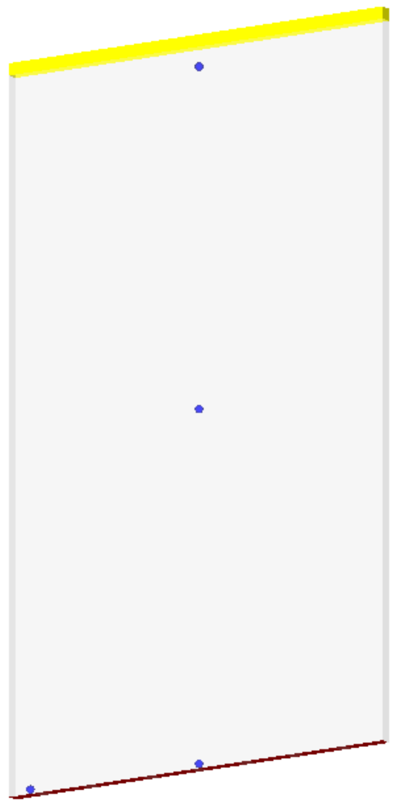
- Click File → New Layout
- Click File → Load Barracuda Data…
- Click Load Data File
- Select trans.data00
- Click Open

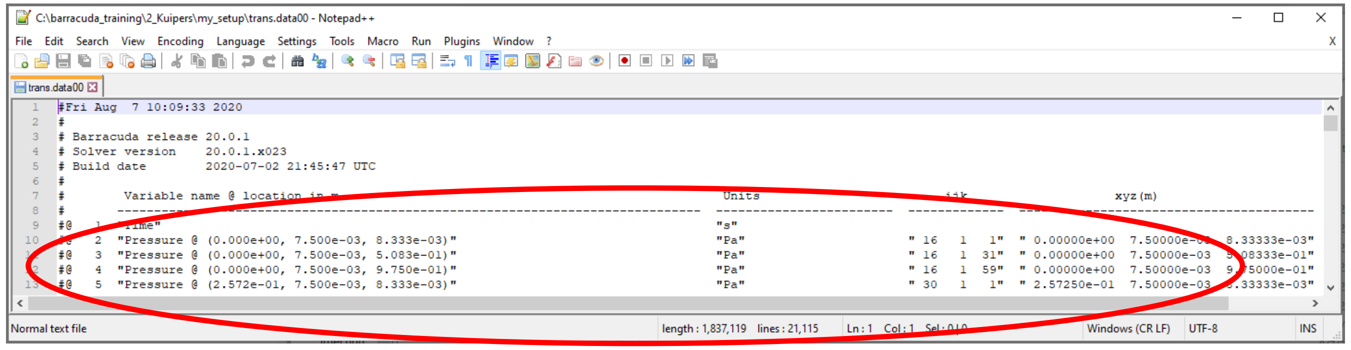


Data Points File Contents

The file trans.data00 header text lists the data collected in each column of the file:

- Column 1 corresponds to Time
- Column 2 corresponds to Pressure at cell (16, 1, 1)
- Column 3 corresponds to Pressure at cell (16, 1, 31)
- Column 4 corresponds to Pressure at cell (16, 1, 59)
- Column 5 corresponds to Pressure at cell (30, 1, 1)



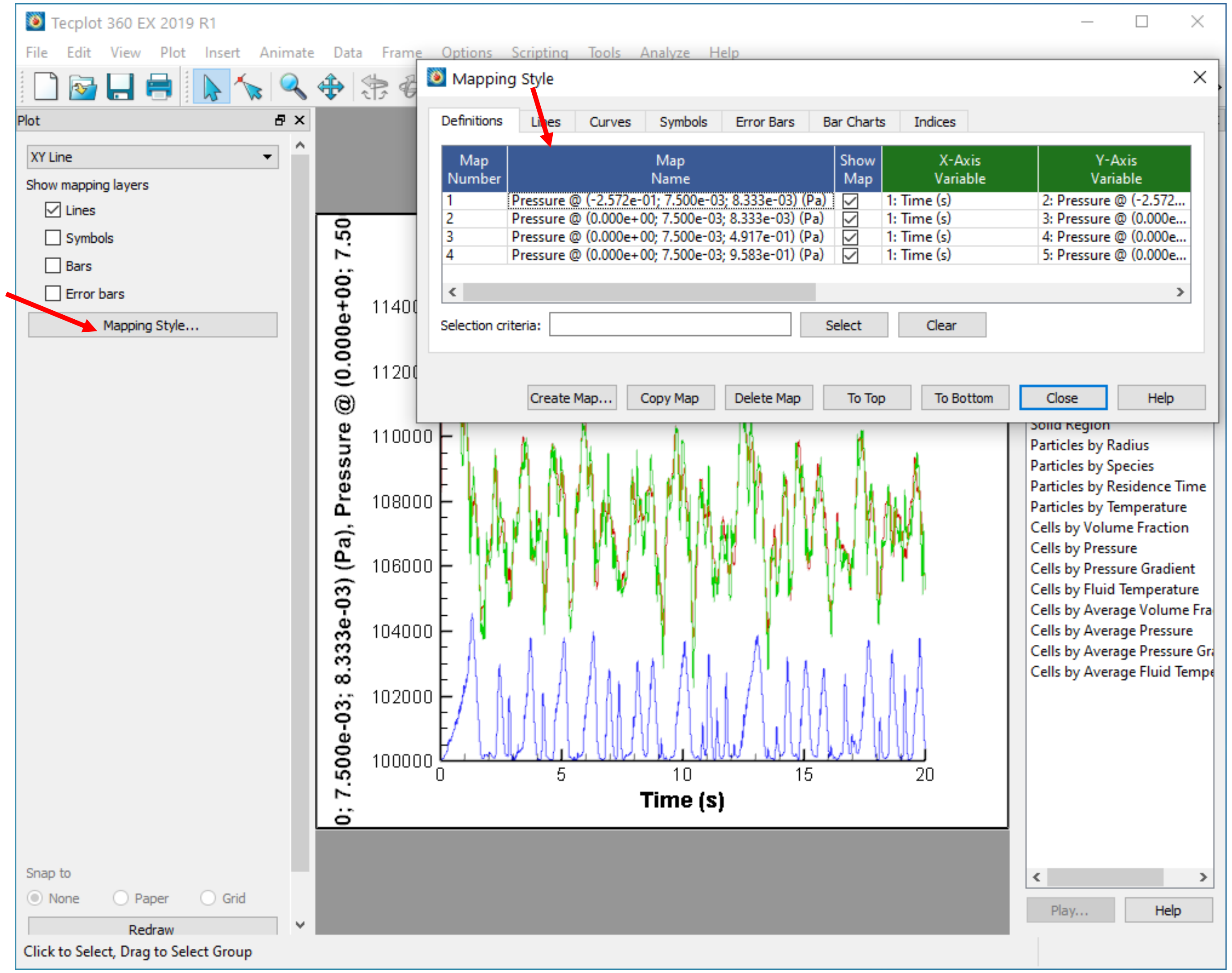



```
#Fri Aug  7 10:09:33 2020
#
# Barracuda release 20.0.1
# Solver version    20.0.1.x023
# Build date        2020-07-02 21:45:47 UTC
#
#        Variable name @ location in m                                 Units                  ijk              xyz (m)
#       -----------------------------------------------------------    ----------------   ------------   ---------------------------------------------
#@   1  "Time"                                                         "s"
#@   2  "Pressure @ (0.000e+00, 7.500e-03, 8.333e-03)"                 "Pa"               " 16   1   1"   " 0.00000e+00  7.50000e-03  8.33333e-03"
#@   3  "Pressure @ (0.000e+00, 7.500e-03, 5.083e-01)"                 "Pa"               " 16   1  31"   " 0.00000e+00  7.50000e-03  5.08333e-01"
#@   4  "Pressure @ (0.000e+00, 7.500e-03, 9.750e-01)"                 "Pa"               " 16   1  59"   " 0.00000e+00  7.50000e-03  9.75000e-01"
#@   5  "Pressure @ (2.572e-01, 7.500e-03, 8.333e-03)"                 "Pa"               " 30   1   1"   " 2.57250e-01  7.50000e-03  8.33333e-03"
```

Plotting Data Points

Click the Mapping Style… button

- Go to the Definitions tab
- Select the boxes in the Show Map column for all four pressure data points
- Ctrl+F will autoscale the axes



Editing the Plot Legend

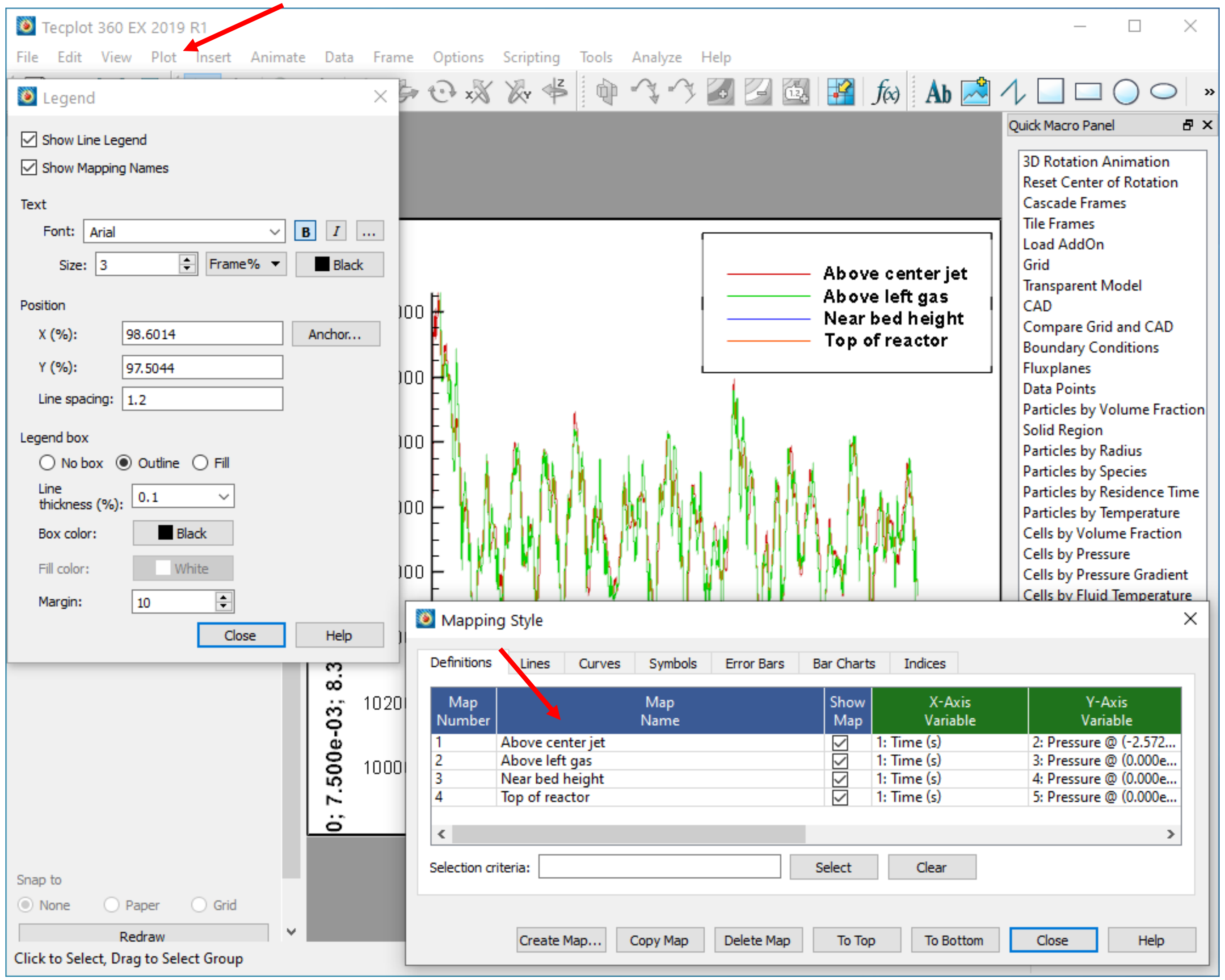
Navigate to Plot → Line Legend…

- Select Show Line Legend to display the legend

In Legend dialog, you can edit other aspects of the legend's appearance

To change the names of the lines in the legend, you must change the names of the variables in the Mapping Style dialog

- Double-click the Map Name for each line to edit the text

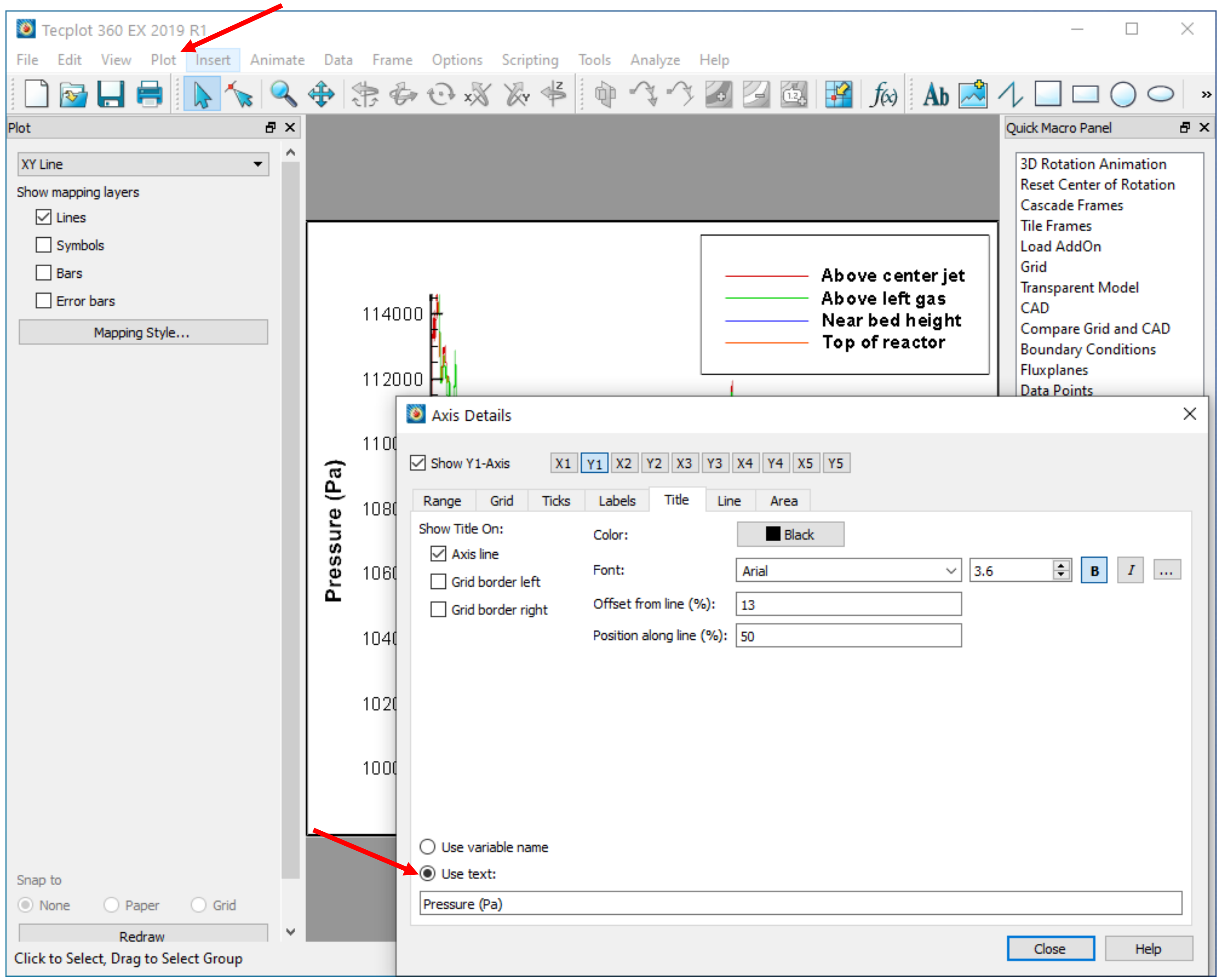


Editing the Axis Labels

Navigate to Plot → Axis…

- Go to the Title tab
- Select Use text
- Enter a custom axis title
- Click Close

The Axis Details window allows you to edit other aspects of axis appearance

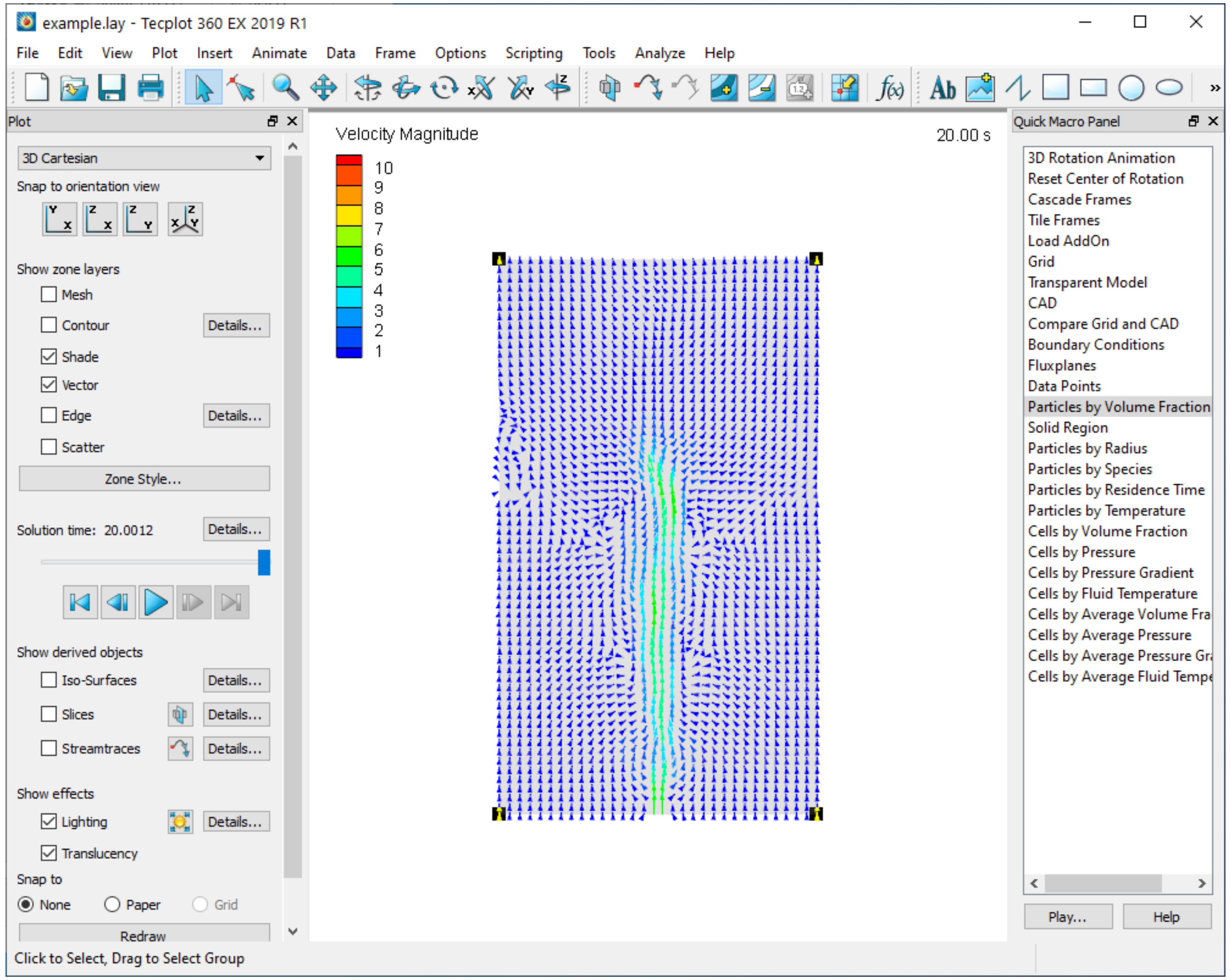


Vectors Showing Instantaneous Fluid Velocity

In this section, we will show how to create a Vector map of the domain in Tecplot

Like Contour Maps and Particle Maps, Vectors are their own zone layer.

Follow the [Creating Vectors] video, to create your own vector plot.

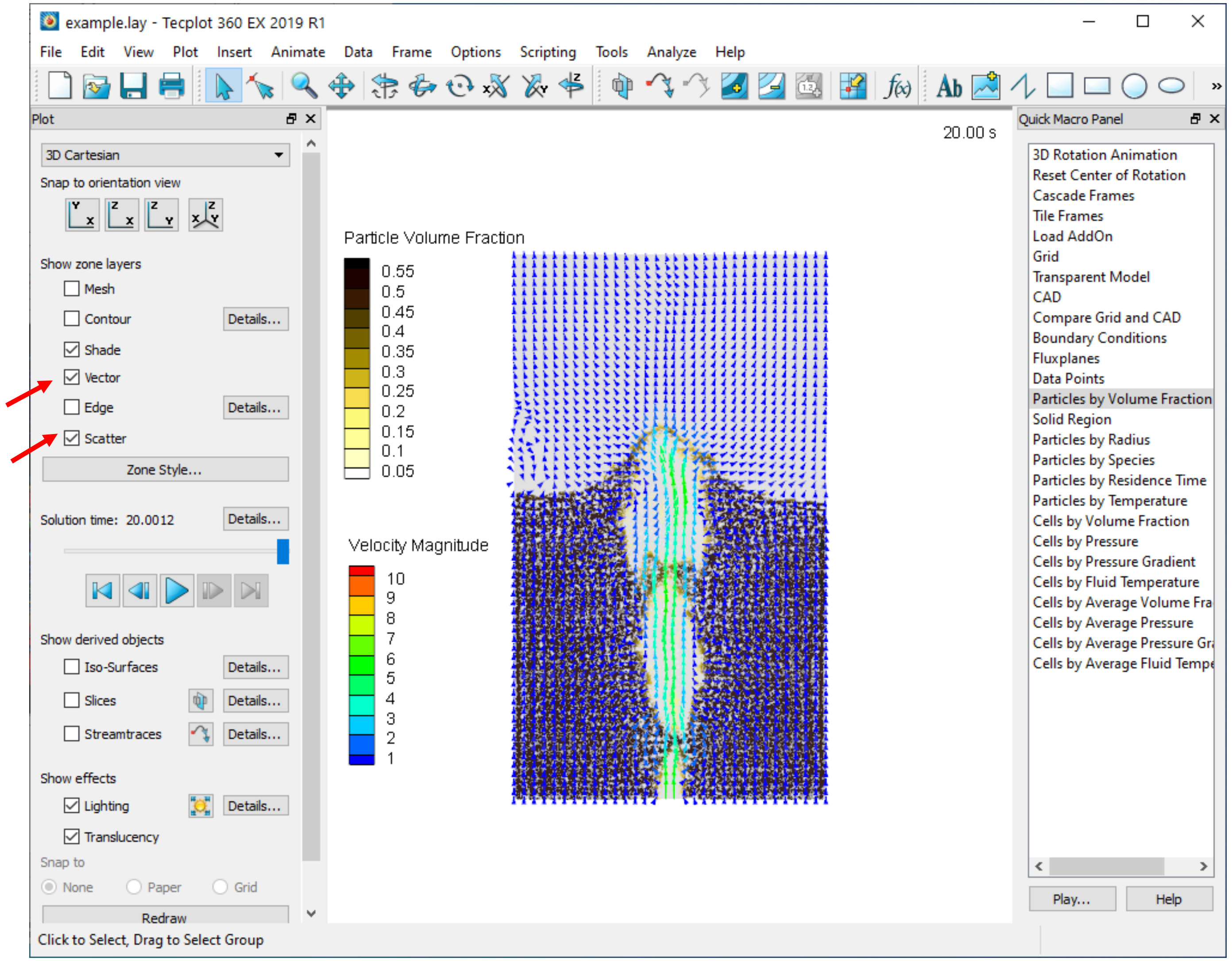


Displaying Vectors and Particles

Multiple zone layers can be toggled on simultaneously in Tecplot

To see both particles and vectors, simply enable the Scatter and Vector layers in the Plot sidebar

Each layer can be controlled and modified independently



Post-Processing Assignment: Average Particle Mass Flux

Create these images of the time-average particle mass flux in the z-direction

How do they provide additional information on particle movement in the bed?