

# Kuipers Bed Training Problem

February 2018

CPFD Software LLC  
10899 Montgomery Blvd. NE, Suite A  
Albuquerque, NM 87111  
+1.505.275.3849  
[www.cdfd-software.com](http://www.cdfd-software.com)

# Training Goals

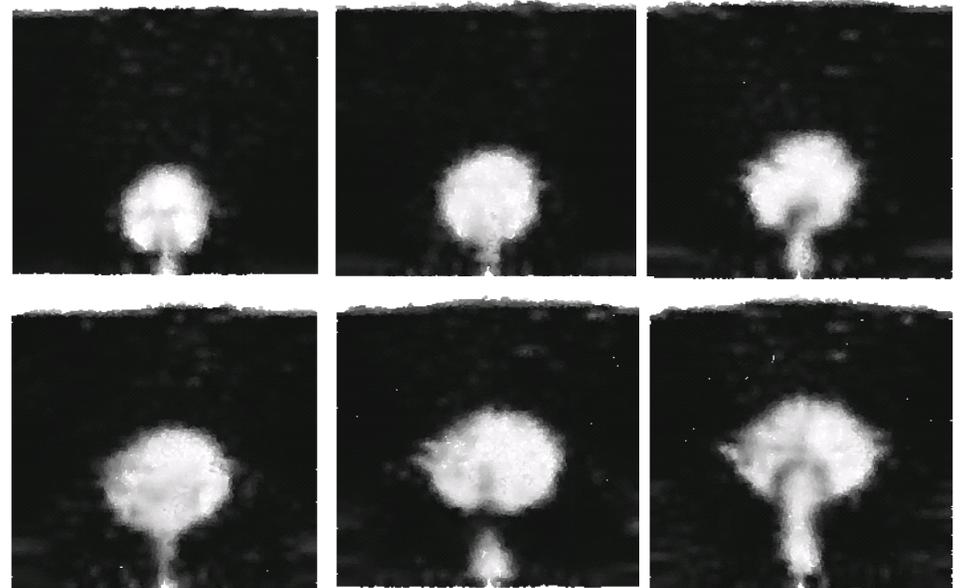
- This training example is designed to provide a quick introduction to the overall problem set-up process. Explanations are minimal. The Barracuda Graphical User Interface (GUI) will be explained in more detail later in this training class. The purpose of the assignments during the first day is to give the user an overall feel for the process and tools involved.

# Training Objectives

- Launch Barracuda
- Open project file with pre-setup grid
  - (Advanced gridding to be discussed during second day of training)
- Set up your calculation with the information provided
- Successfully run Kuipers Bed simulation
- Ask your instructor about any questions you may have

# The Kuipers Bed

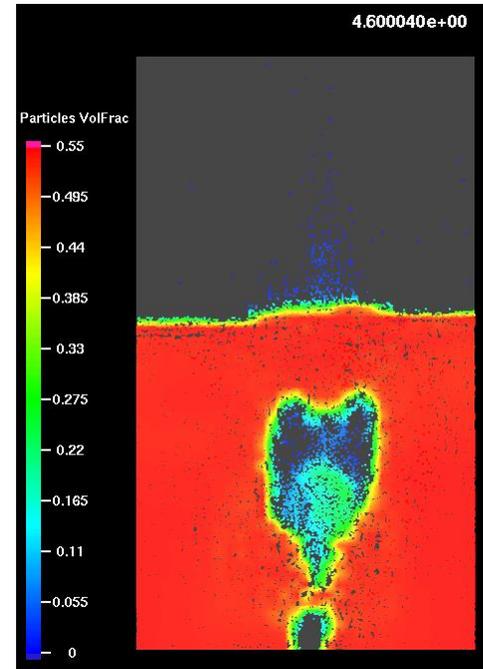
- Kuipers et al. (1992) operated experimental “2D” fluidized bed
- The experimental setup consisted of
  - A thin rectangular bed (.57m x1.0m x 1.5 cm)
  - Fluidizing gas (0.25 m/s velocity) on bottom of reactor
  - Central gas jet (10 m/s)
  - Open to atmosphere at top
- During operation, Kuipers observed and measured
  - Bubbling behavior of bed
  - Bubble diameter
  - Particle volume fraction at various locations using IR sensor probe
- First training problem is to simulate the Kuipers bed with Barracuda



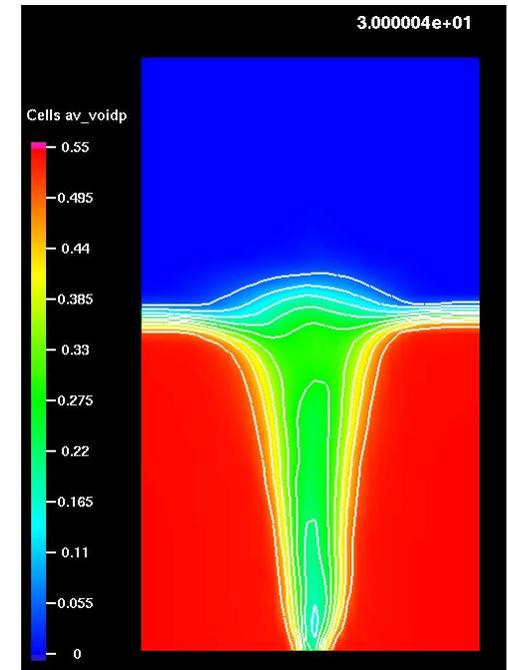
Kuipers, J, Tammes, Prins, and Swaaij (1992). Powder Technology 71: 87-99

# Simulation Objectives

- What can we learn from the simulation?
  - Fluidization behavior
  - Bubble diameters
  - Particle volume fraction
  - Particle mixing in time
  - Velocity data
    - Fluid
    - Particle
  - Pressure profiles
  - Pressure gradients

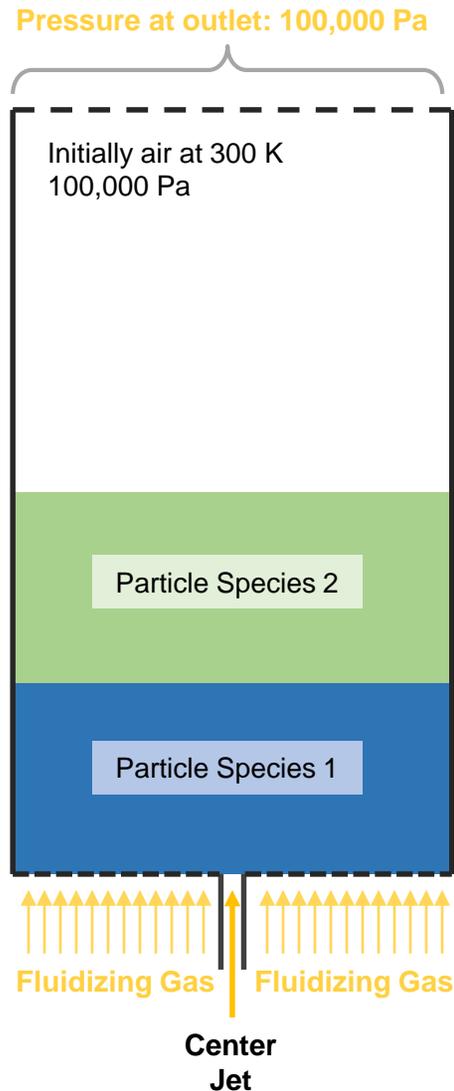


Instantaneous  
particle volume fraction



Time averaged  
particle volume fraction

# Process Sheet



- Geometry
  - 57 cm wide
  - 1.5 cm deep
  - 100 cm total height
  - 50 cm initial bed height
  - 1.5 cm X 1.5 cm jet centered at bottom
- Particles
  - Use two identical particle species to view mixing behavior
  - Material density 2.66 g/cm<sup>3</sup> (glass beads)
  - Diameter: 440µm - 560µm. This is 500 µm ±12%
- Initial conditions
  - Fluid phase: air at rest at atmospheric pressure
  - Solid phase: particles in bottom ½ of bed at close-pack ( $\theta_{cp} = 0.55$ )
- Boundary conditions
  - Fluid
    - Grid velocity: 0.25 m/s
    - Center jet velocity: 10 m/s
    - Top open to atmosphere
  - Particles
    - Cannot enter or leave

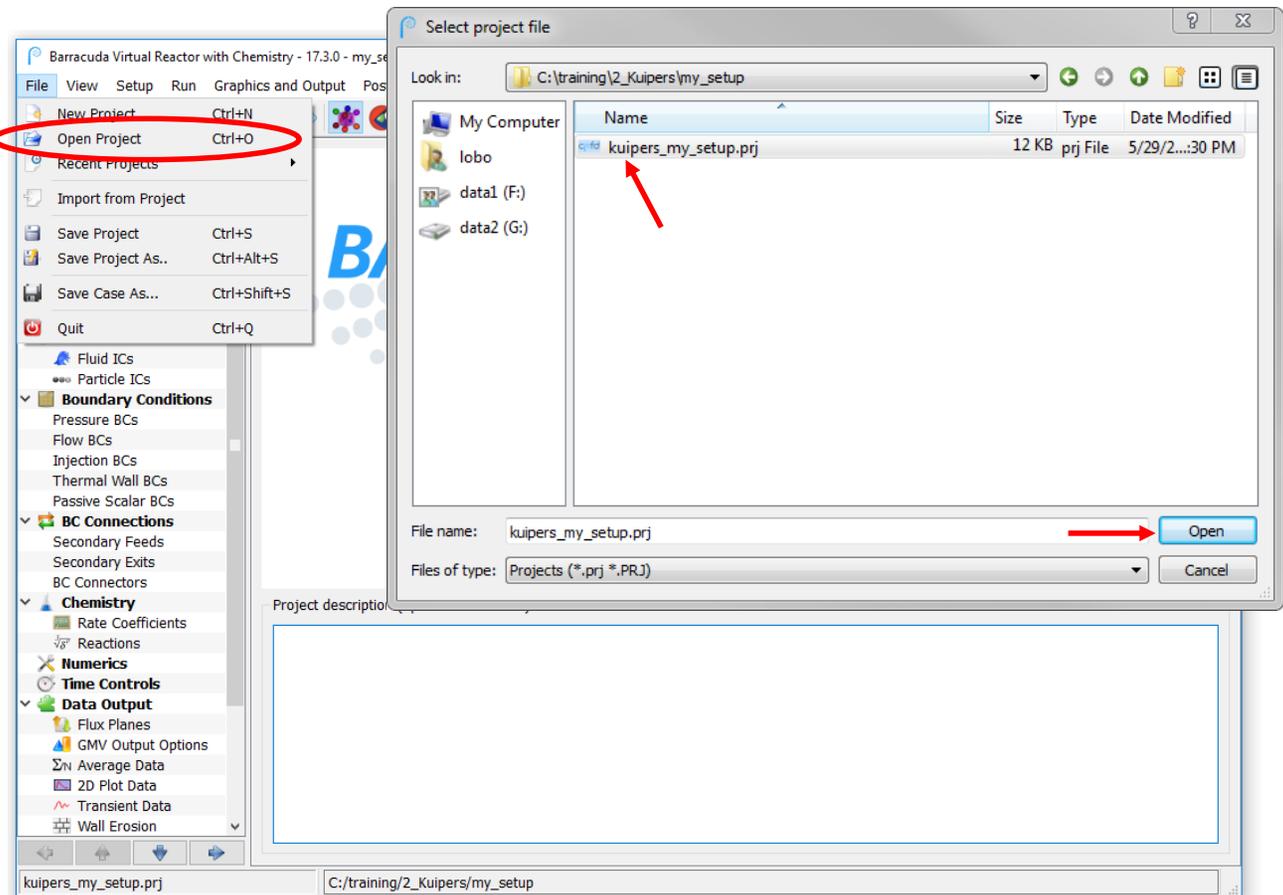
# Launching Barracuda

- Launch Barracuda. To do so you may click on the desktop icon or type “barracuda.17” at the command line



# Opening a Project File

- Click on **File**, then **Open Project** to open an existing project.
- A project file window will pop up
- Navigate to:
  - On Linux: /home/training/barracuda\_training/2\_Kuipers/my\_setup
  - On Windows: C:\training\2\_Kuipers\my\_setup
- Select **kuipers\_mysetup.prj**
- Click **Open Project**
- The path to your training directory may be different than shown above. Use the appropriate path when navigating to your project.

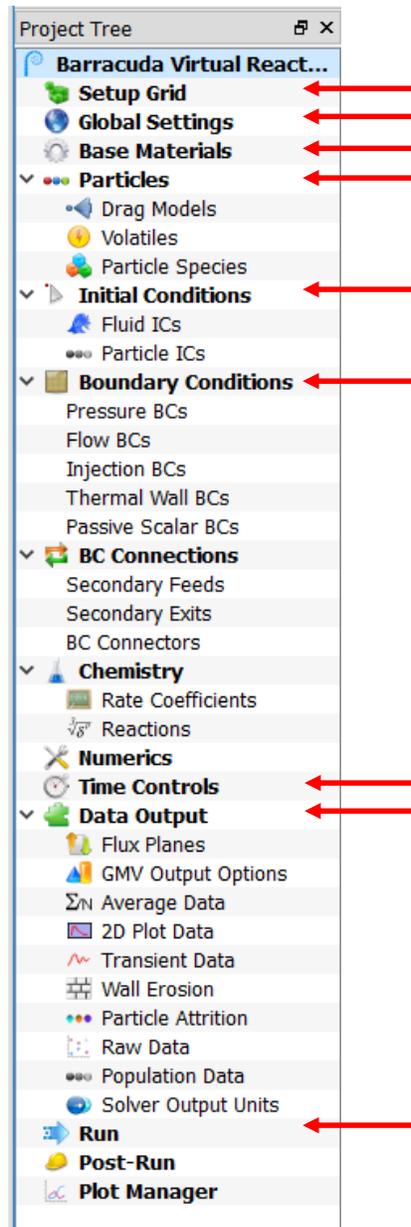


# Navigating the Barracuda GUI

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



# Setting up the Kuipers Bed Simulation



The Kuipers Bed simulation will be set up in 9 steps:

## 1. Setup Grid

Create the computational grid from a CAD file of the physical geometry

## 2. Global Settings

Set gravity and select isothermal calculation

## 3. Base Materials

Add materials to simulation and edit physical properties

## 4. Particles

Specify materials and particle size distributions for the particles in the model

## 5. Initial Conditions

Specify initial fluid and particle conditions. Specify initial particle locations

## 6. Boundary conditions

Specify fluid velocities and pressures at model boundaries

## 7. Time Controls

Specify simulation time, time step and restart interval

## 8. Data Output

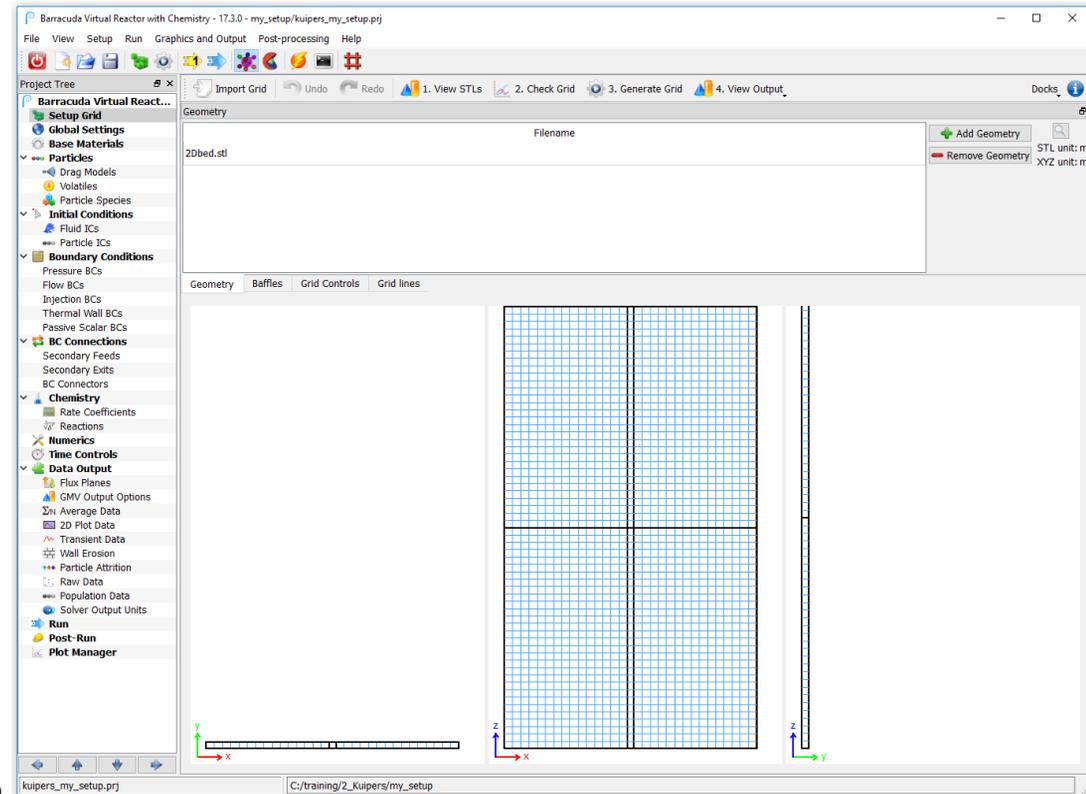
Select data to be written during simulation for later analysis

## 9. Run

Check the model setup and run the simulation

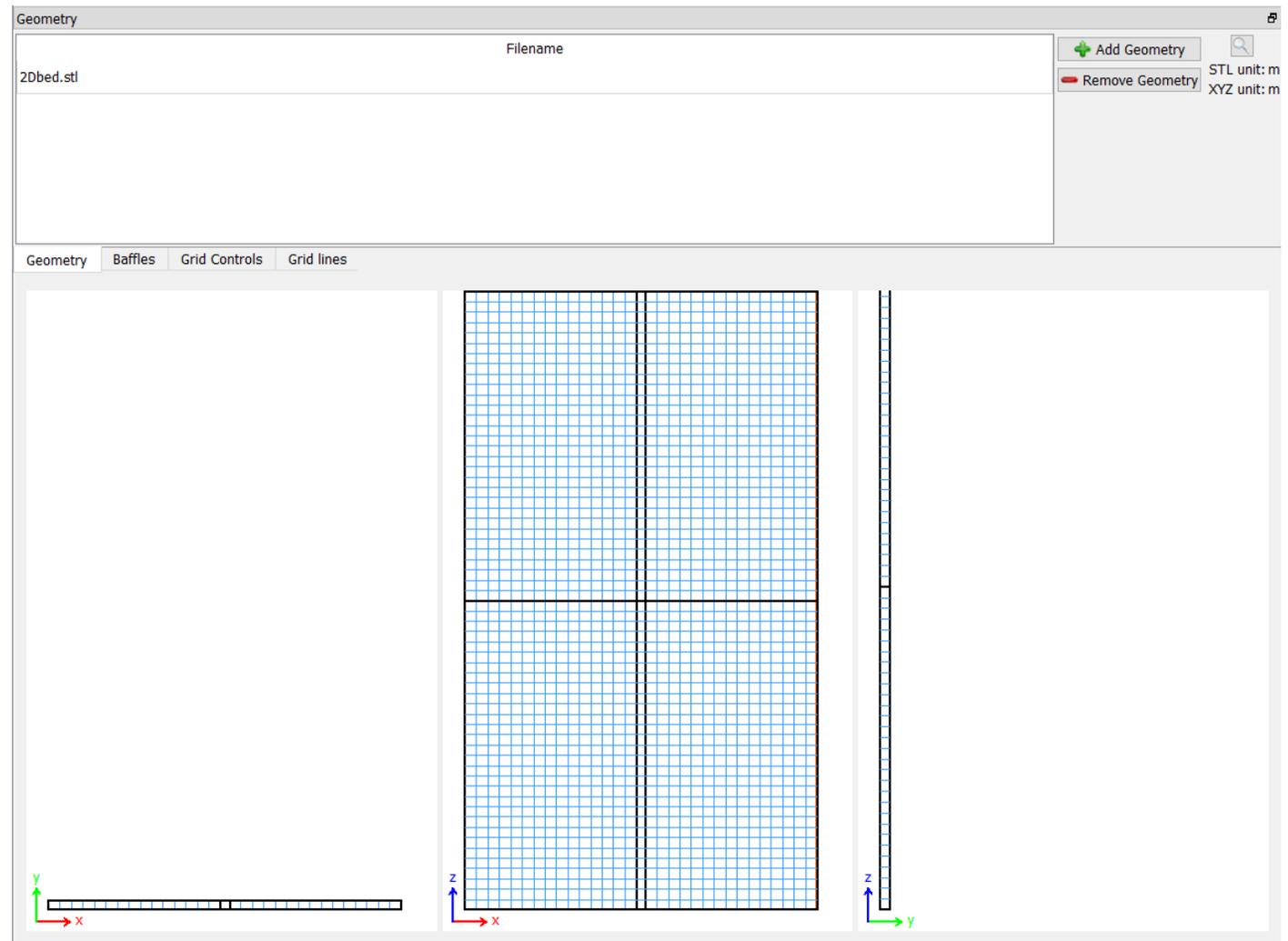
# Setup Grid

- 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
- Adding cells to a simulation will also increase the computational time required
- **For the Kuipers problem, gridline locations are already provided in the kuipers\_mysetup.prj file**



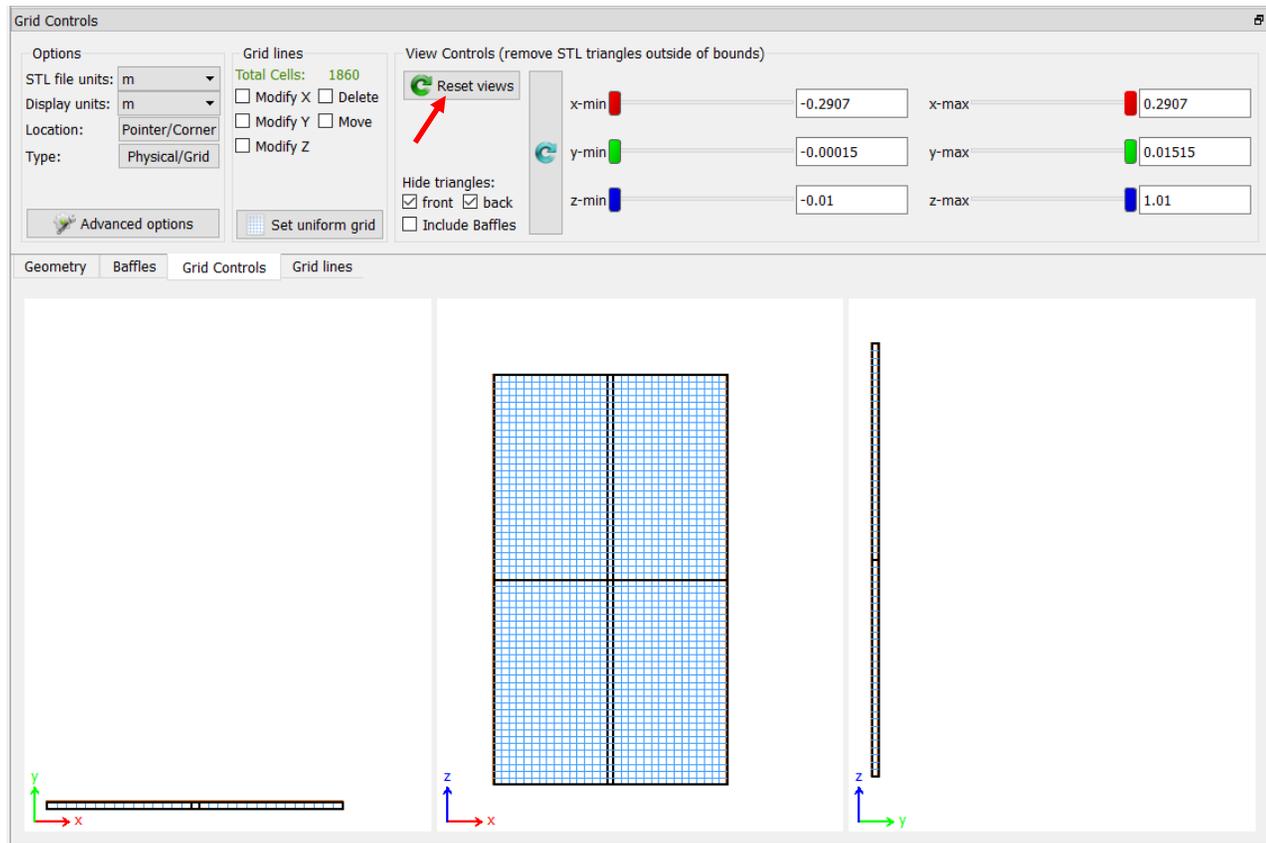
# Setup Grid Window

- The setup grid window provides an interface for creating an appropriate grid for the model
- It contains the following tabs:
  - Geometry
  - Baffles
  - Grid Controls
  - Grid lines



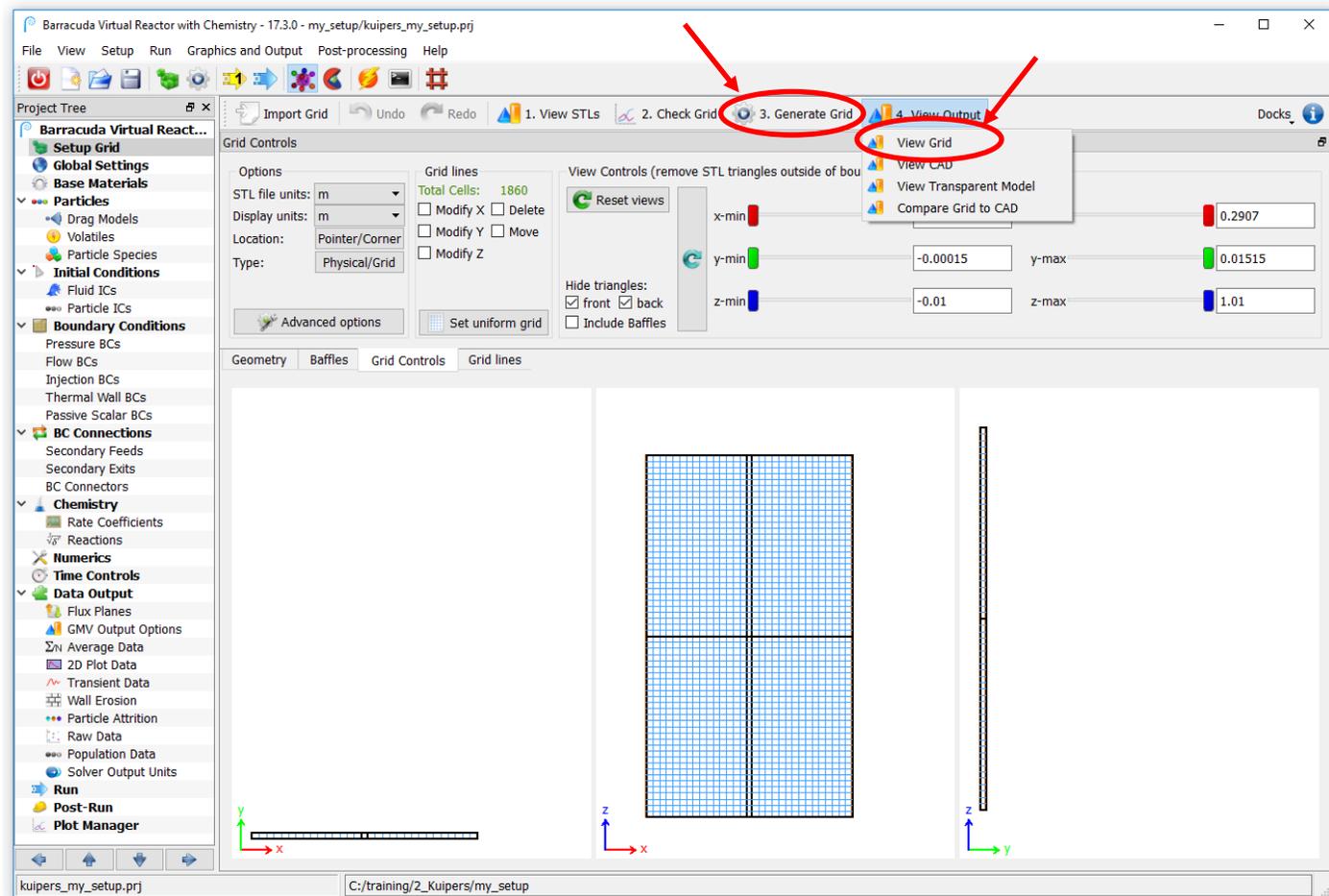
# Grid Controls

- 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 (pending hardware and software configuration)
  - Note: each of the panel views can be panned and zoomed independently
- GUI button resets the grid view



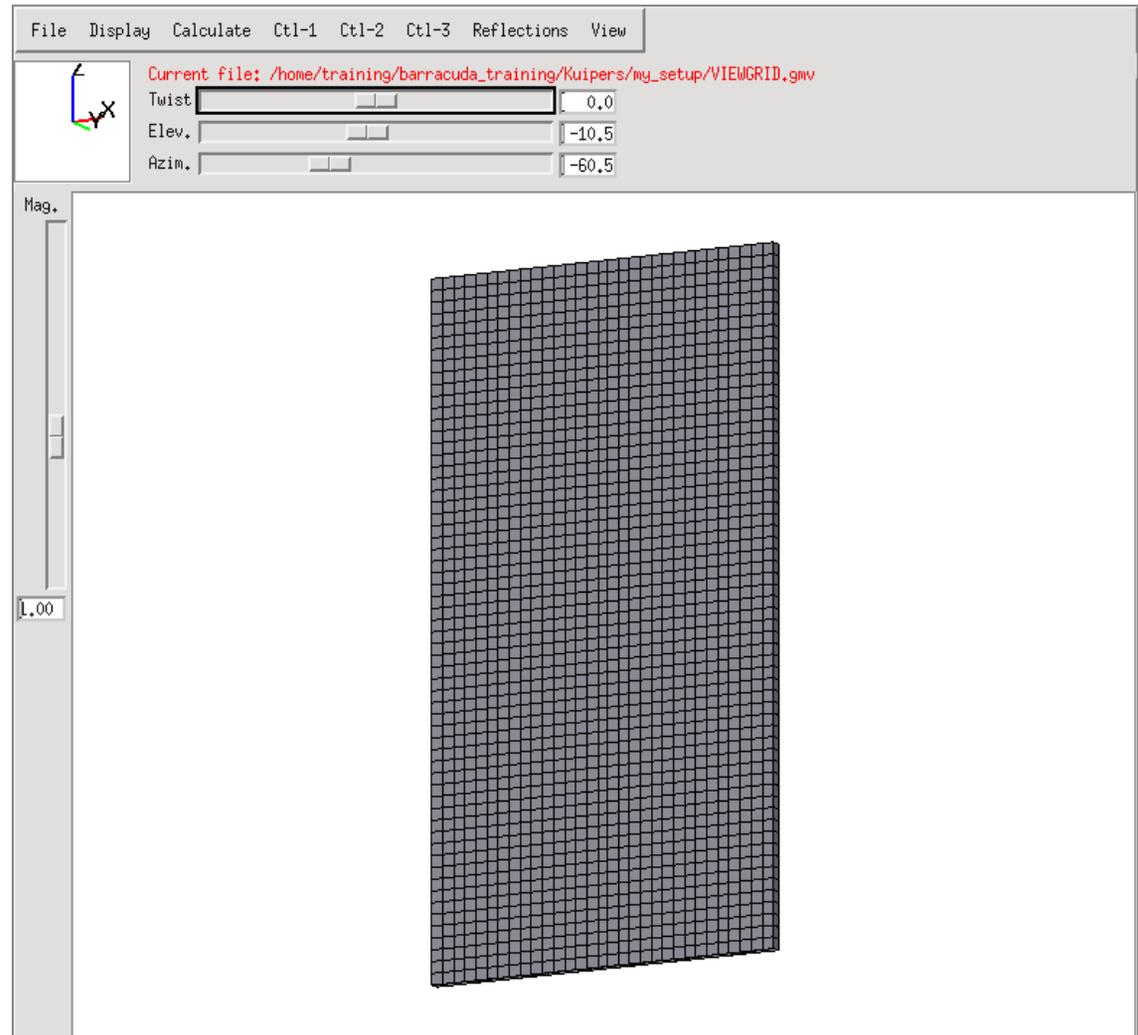
# Generating and Viewing the Grid

- The computational grid will be generated based on the gridlines and STL file
- Click **Generate Grid**
- Once the grid generator runs, open the grid using the **View Output** button and selecting **View Grid**



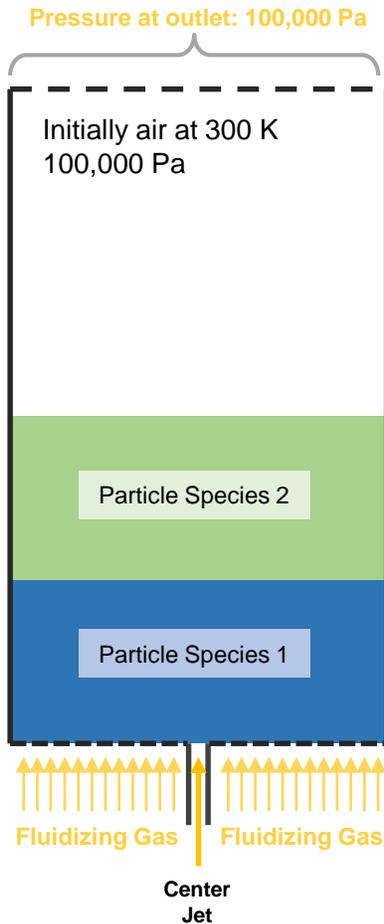
# Viewing the Grid

- The grid may be rotated using the left mouse button
- To translate the grid, use the center mouse button
- Zooming is accomplished using the right mouse button
- **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).

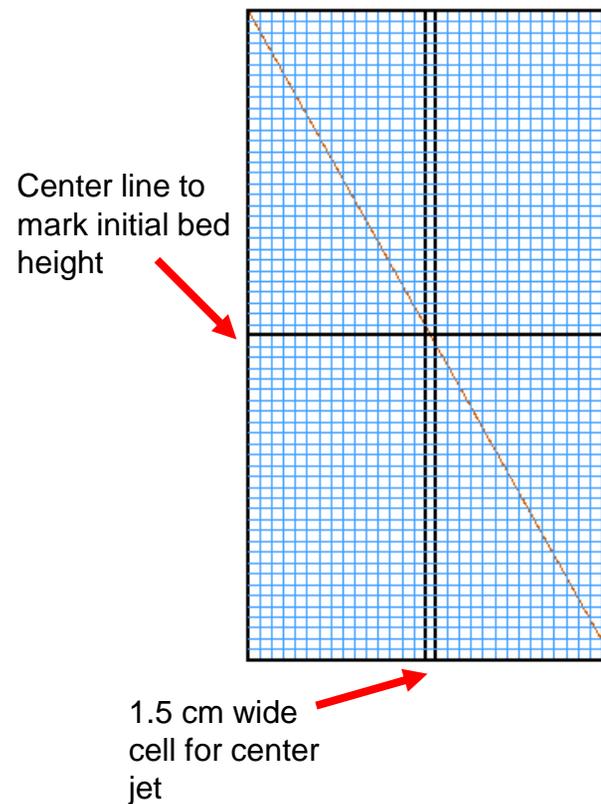


# Overview of the Kuipers Bed Grid

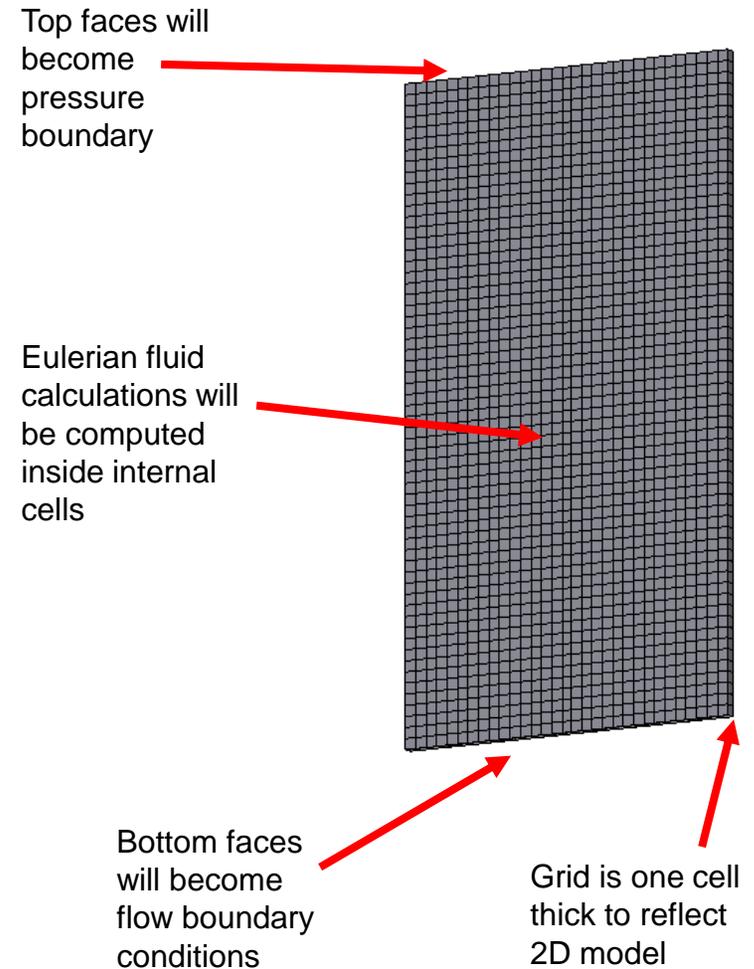
## Physical Setup



## Grid Setup



## Generated Grid



# Grid Coordinates

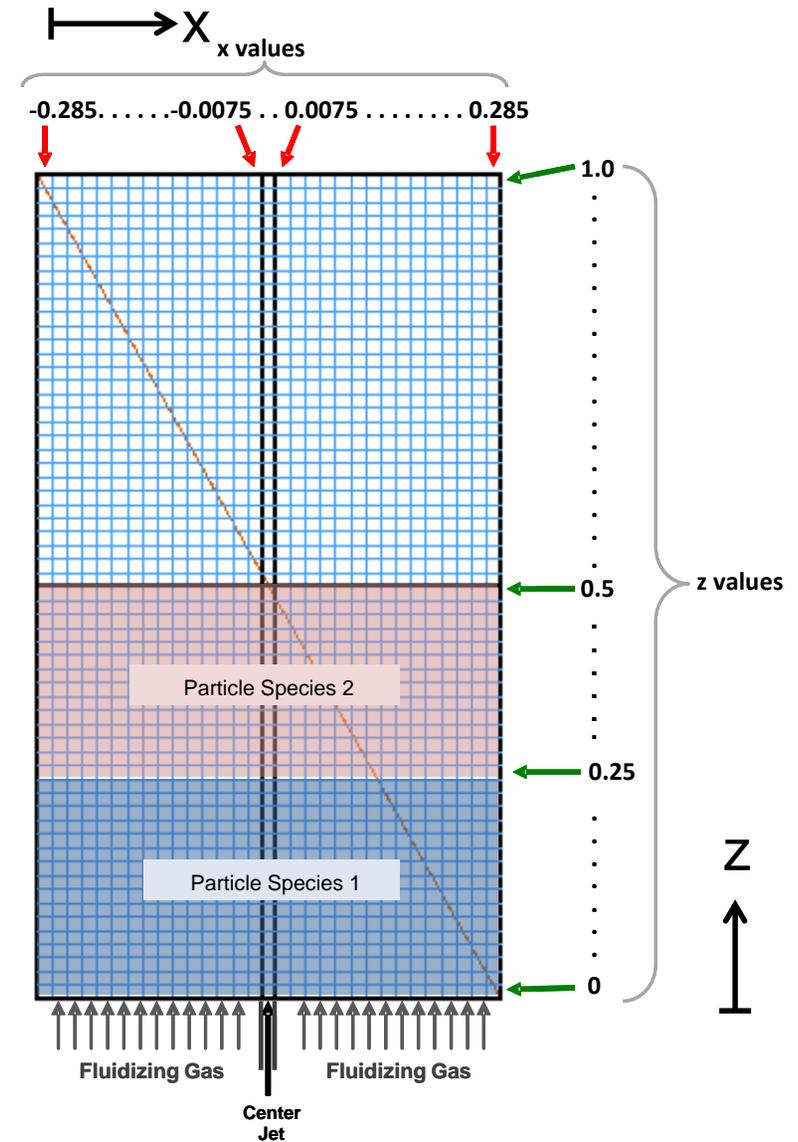
- Initial and Boundary Conditions are located by x,y,z values
- [ uses the minimum value, ] uses the maximum value

## Initial Particle Location:

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

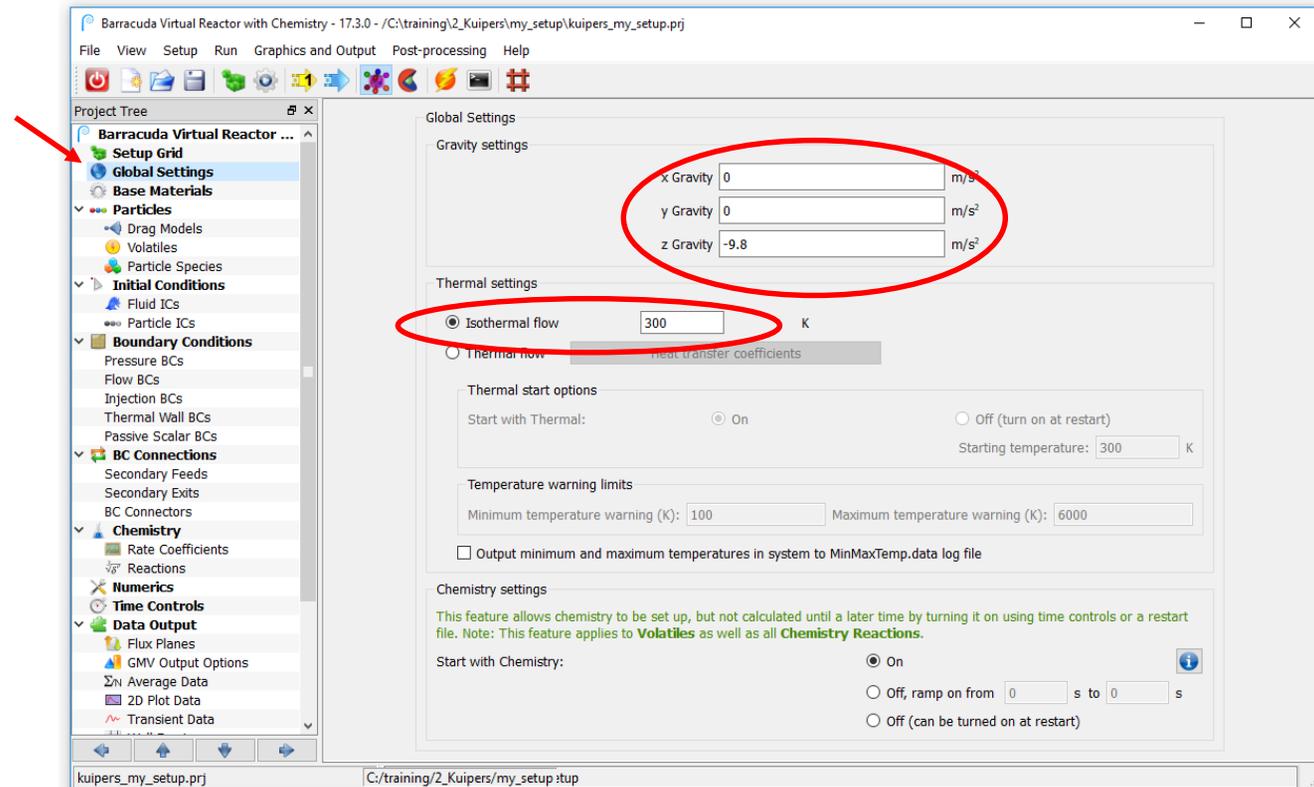
## Boundary Condition Location:

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



# Global Settings

- Click on **Global Settings**
- Set your **Gravity** vector
  - In this example, gravity is in the negative z-direction
  - Notice the vector magnitude is “9.8” m/s<sup>2</sup>
  - Note: All units in Barracuda are SI unless otherwise noted
- Isothermal flow should be selected.

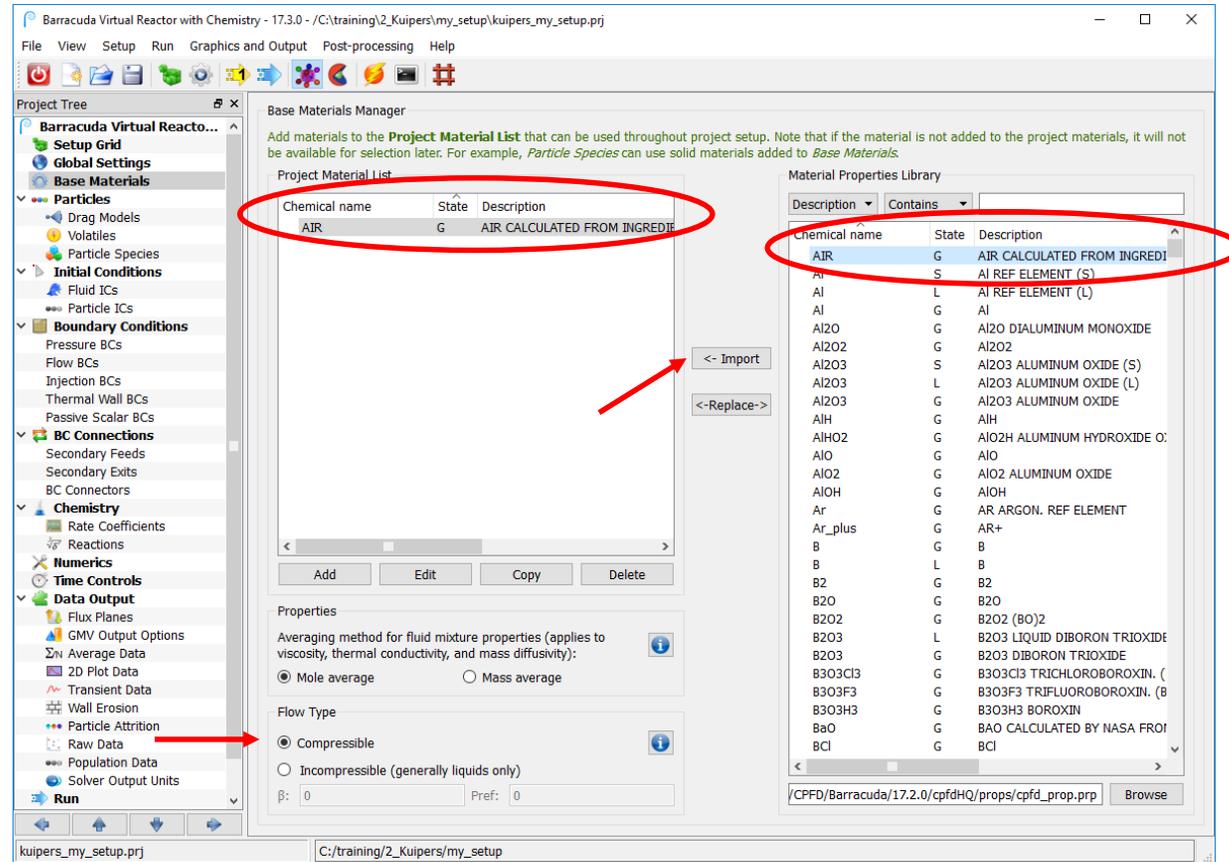




# Base Materials

For the Kuipers setup:

- Verify **Compressible flow** is selected, since the fluid is a gas
- Air and glass need to be added
- Select AIR from the **Material Library**
- Click **Import**
- AIR now appears in the **Project Material List** at the left



# Base Materials

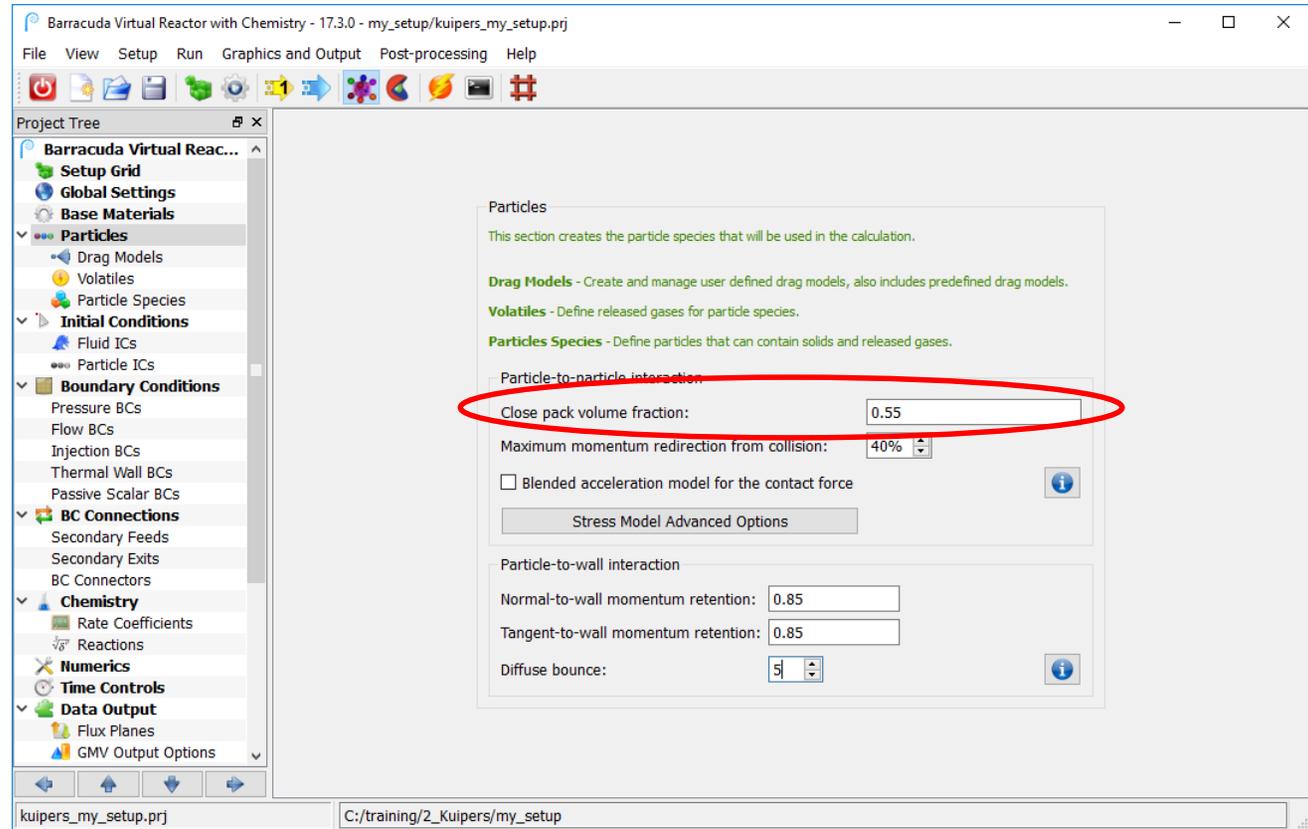
- To define the solid material, click on **Add**
- In the **Material Properties** window, enter the **Chemical name** as "GLASS". Enter an appropriate description
- Set the **State (Gas, Liquid, Solid)** to solid
- Set the **Density** to  $2660 \text{ kg/m}^3$
- Set the **Molecular weight** to  $60 \text{ g/mol}$
- Click **OK**
- GLASS now appears in the **Project Material List** at the left

The screenshot shows the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The "Project Tree" on the left shows the "Base Materials" folder expanded, with "GLASS" listed under "Project Material List". The "Base Materials Manager" window is open, displaying the "Project Material List" with the following description: "AIR CALCULATED FROM INGREDIENTS %N2=78.084 %O2=20.9 GLASS. SODA-LIME SILICA. COMMON FOR GLASS BEADS". The "Base Materials Editor" window is also open, showing the "Name" field set to "GLASS", "State" set to "Solid", "Molecular weight" set to "60 g/mol", and "Density" set to "2660 kg/m³". The "Base Materials Editor" window has "OK" and "Cancel" buttons at the bottom right. The "Base Materials Editor" window also shows a list of materials in the background, including AIOH, Ar, Ar\_plus, B, B2, B2O, B2O2, B2O3, B2O3, B2O3Cl3, B3O3Cl3, B3O3F3, B3O3H3, BaO, and BCl.

- **NOTE:** 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
- **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

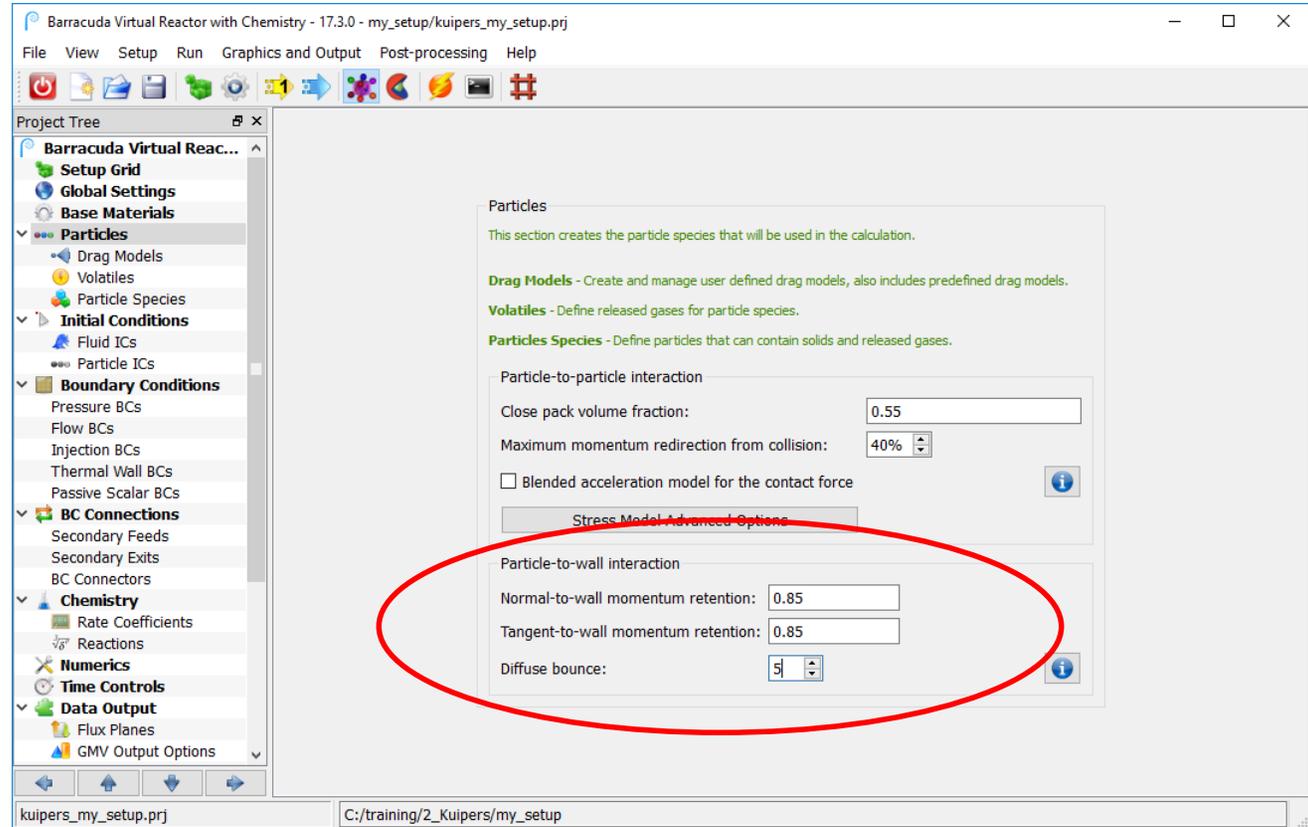
# Particles

- Click on the **Particles** page
- Enter a **Close pack volume fraction** of “0.55”
- This is the maximum amount of solids packing permitted in a region
- 0.55 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

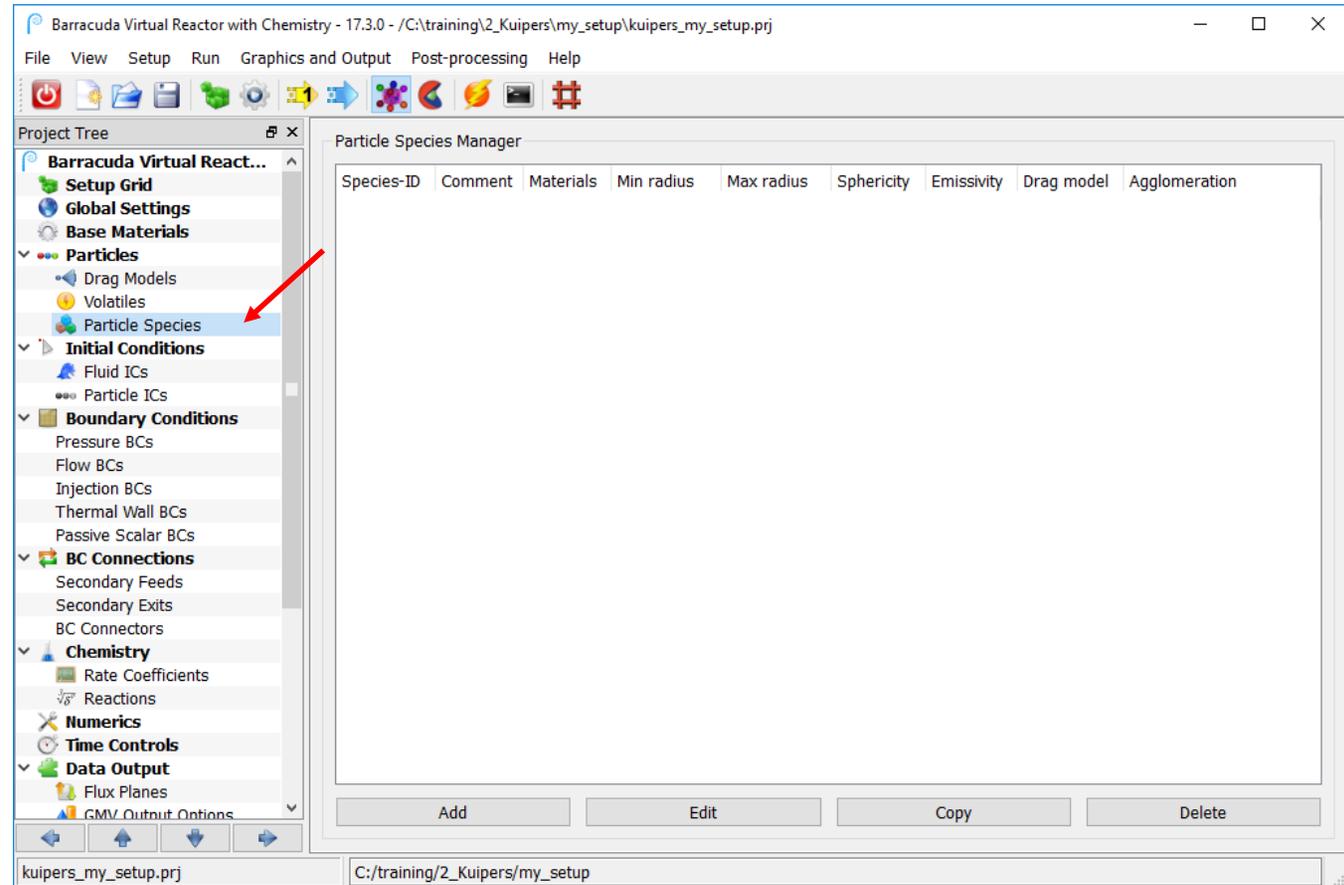


# Particle Species

- Click on **Particle Species**
- We are now ready to define the particle species in the system

## For the Kuipers setup:

- 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

- Click on **Add**
- The **Particle Species Editor** window will pop up, click on **Applied Materials**
- **Applied Materials Manager** window will pop up, click on **Add**
- In the **Applied material** window, select GLASS
- Set the **Mass fraction amount** to “1”
- Click **OK**
- Click **OK** in **Applied Materials Manager** window.

The screenshot displays the Barracuda Virtual Reactor software interface. The **Particle Species Editor** window is open, showing the **Materials** tab with **Applied Materials** selected. The **Applied Materials Manager** window is also open, showing the **Applied material** dialog box. In this dialog, the **Mass fraction amount** is set to 1. The **Project material list** shows **GLASS S** selected. The **OK** button is highlighted. The **Applied material** dialog box is also open, showing the **Mass fraction amount** set to 1. The **OK** button is highlighted. The **Applied material** dialog box is also open, showing the **Mass fraction amount** set to 1. The **OK** button is highlighted.

# Particle Species

- We are now ready to define the particle radius
- For the Kuipers setup, the particle diameters are between 440 and 560  $\mu\text{m}$
- Set the particle **Radius** to “2.2e-4 to 2.8e-4” meters
- Since we have two identical particle species, add a descriptive **Comment** for species 1
- Select Wen-Yu as **Drag model** in the drop down menu
- Click **OK**

The screenshot shows the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - my\_setup/kuipers\_my\_setup.prj". The Project Tree on the left shows the setup configuration, with "Particle Species" selected. The Particle Species Manager window is open, showing a table of species. The Particle Species Editor window is also open, showing the configuration for species 1. The Species-ID is 1, the Comment is "glass beads (bottom of vessel)", and the Drag Model is "Wen-Yu". The Radius is set to a range from 2.2e-04 to 2.8e-04 meters. The PSD File name is empty. The Sphericity is 1, and the Emissivity is 1. The Multiplier (constant) is 1. The Radius Cut Point is 1.8e-05 meters. The Effective Particle Size File name is empty. The OK button is highlighted with a red arrow.

Species-ID	Comment	Materials	Min radius	Max radius	Sphericity	Emissivity	Drag model	Agglomera
1	glass beads (bottom of vessel)	Applied Materials					Wen-Yu	

Name	Link To Default	Value
c0	<input checked="" type="checkbox"/> Linked	1
c1	<input checked="" type="checkbox"/> Linked	0.15
c2	<input checked="" type="checkbox"/> Linked	0.44
n0	<input checked="" type="checkbox"/> Linked	-2.65

# Particle Species

- To define a second, identical particle species, simply copy the first
- Select species **001**, then click **Copy**

The screenshot shows the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The interface includes a menu bar (File, View, Setup, Run, Graphics and Output, Post-processing, Help) and a toolbar with various icons. On the left is a "Project Tree" with a hierarchical view of the simulation setup, including sections like "Global Settings", "Base Materials", "Particles", "Initial Conditions", "Boundary Conditions", "BC Connections", "Chemistry", "Numerics", "Time Controls", and "Data Output". The "Particles" section is expanded, showing "Particle Species" selected. The main area is the "Particle Species Manager" window, which contains a table with the following data:

Species-ID	Comment	Materials	Min radius	Max radius	Sphericity	Emissivity	Drag model	Agglor
001	glass beads (bottom of vessel)	GLASS	2.200000e-04	2.800000e-04	1	1	Wen-Yu	Off

At the bottom of the "Particle Species Manager" window, there are four buttons: "Add", "Edit", "Copy", and "Delete". A red arrow points to the "Copy" button. Another red arrow points to the "001" species ID in the table.

# Particle Species

- To change the comment describing the second particle, select species **002**
- Click **Edit**
- Change the Comment
- Click **OK**

The screenshot displays the Barracuda Virtual Reactor interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - my\_setup/kuipers\_my\_setup.prj". The "Particle Species Manager" window is open, showing a table of particle species. The second species, ID 002, is selected. The "Particle Species Editor" window is also open, showing the details for species 002. The "Comment" field is highlighted with a red circle, and the "OK" button is also highlighted with a red circle. A red arrow points to the "Edit" button at the bottom of the Particle Species Manager window.

Species-ID	Comment	Materials	Min radius	Max radius	Sphericity	Emissivity	Drag model	Agglomera
001	glass beads (bottom of vessel)	Glass	2.200000e-04	2.800000e-04	1	1	Wen-Yu	Off
002	glass beads (top of vessel)	Glass	2.200000e-04	2.800000e-04	1	1	Wen-Yu	Off

Particle Species Editor details:

- Species-ID: 002
- Comment: glass beads (top of vessel)
- Materials: Applied Materials
- Particle Size: Predefined PSD: [dropdown]
- PSD Filename: [text box]
- Radius: Minimum: 0.00022, Maximum: 0.00028 m
- Sphericity: 1
- Emissivity: 1
- Drag Model: Wen-Yu
- Multiplier (constant): 1
- Agglomeration: [checkbox]
- Radius Cut Point: 1.8e-05 m

# Fluid Initial Conditions

- Click on **Fluid ICs**
- Select the default fluid species
- Click on **Edit**
- Click on **Define fluids**

The screenshot displays the Barracuda Virtual Reactor interface. The Project Tree on the left shows the 'Initial Conditions' folder expanded, with 'Fluid ICs' selected. The Fluid IC Manager table shows a single entry with ID 000, which is checked and has a red arrow pointing to it. The 'Edit' button at the bottom of the manager has a red arrow pointing to it. The Fluid IC Editor dialog box is open, showing the 'Initial Conditions' section with fields for Temperature (300 K), Pressure (0 Pa), Fluid species (with a 'Define fluids' button), and Velocity (0, 0, 0 m/s). The 'Region' section shows a 'Select region (m)' button and coordinate fields for x1, x2, y1, y2, z1, and z2. A red arrow points to the 'Define fluids' button in the dialog.

ID	On	x1	x2	y1	y2	z1	z2	x-Vel	y-Vel
000	<input checked="" type="checkbox"/>	-0.285	0.285	0	0.015	0	1	0	0

# Fluid Initial Conditions

- The **Applied Materials** window will pop up. Click on **Add Material**
- Select **AIR** and set the **Mass fraction amount** to “1”
- Click **OK**

The screenshot displays the Barracuda Virtual Reactor interface. The main window shows the Project Tree on the left, with 'Initial Conditions' selected. The 'Fluid IC Manager' table is visible, showing a single entry for ID 000. Overlaid on this are two dialog boxes: 'Applied materials' and 'Applied material'. The 'Applied materials' dialog has a table with columns 'ID', 'Material', 'State', and 'Fraction'. A red arrow points to the 'Add material' button. The 'Applied material' dialog shows a 'Project material list' with 'AIR' and 'G' listed. A red circle highlights the 'Mass fraction amount' field, which contains the value '1'. Another red arrow points to the 'OK' button in the 'Applied material' dialog. The 'Fluid IC Editor: 000' dialog is also visible in the background, showing fields for Temperature (300 K), Pressure (0 Pa), Fluid species (Define fluids), and Velocity (0 m/s).

ID	On	x1	x2	y1	y2	z1	z2	x-Vel	y-Vel
000	<input checked="" type="checkbox"/>	-0.285	0.285	0	0.015	0	1	0	0

ID	Material	State	Fraction
	AIR	G	

# Fluid Initial Conditions

- Once the fluid is defined, specify the velocity, pressure, and location of the fluid
- In this example problem, air is initially at rest at atmospheric pressure
- Leave the x, y, and z fluid velocities at zero
- Set the **Pressure** to “1e5” Pa
- Since the air initially occupies the entire bed (area above and in between quartz particles), use [ and ] to input the minimum and maximum values
- Click **OK**

Fluid IC Editor: 000

Initial Conditions

Temperature 300 K

Pressure 1e5 Pa

Fluid species Define fluids

Velocity 0 0 0 m/s

Initial conditions from file

IC File

Region

Select region (m)

x<sub>1</sub> -0.285 x<sub>2</sub> 0.285

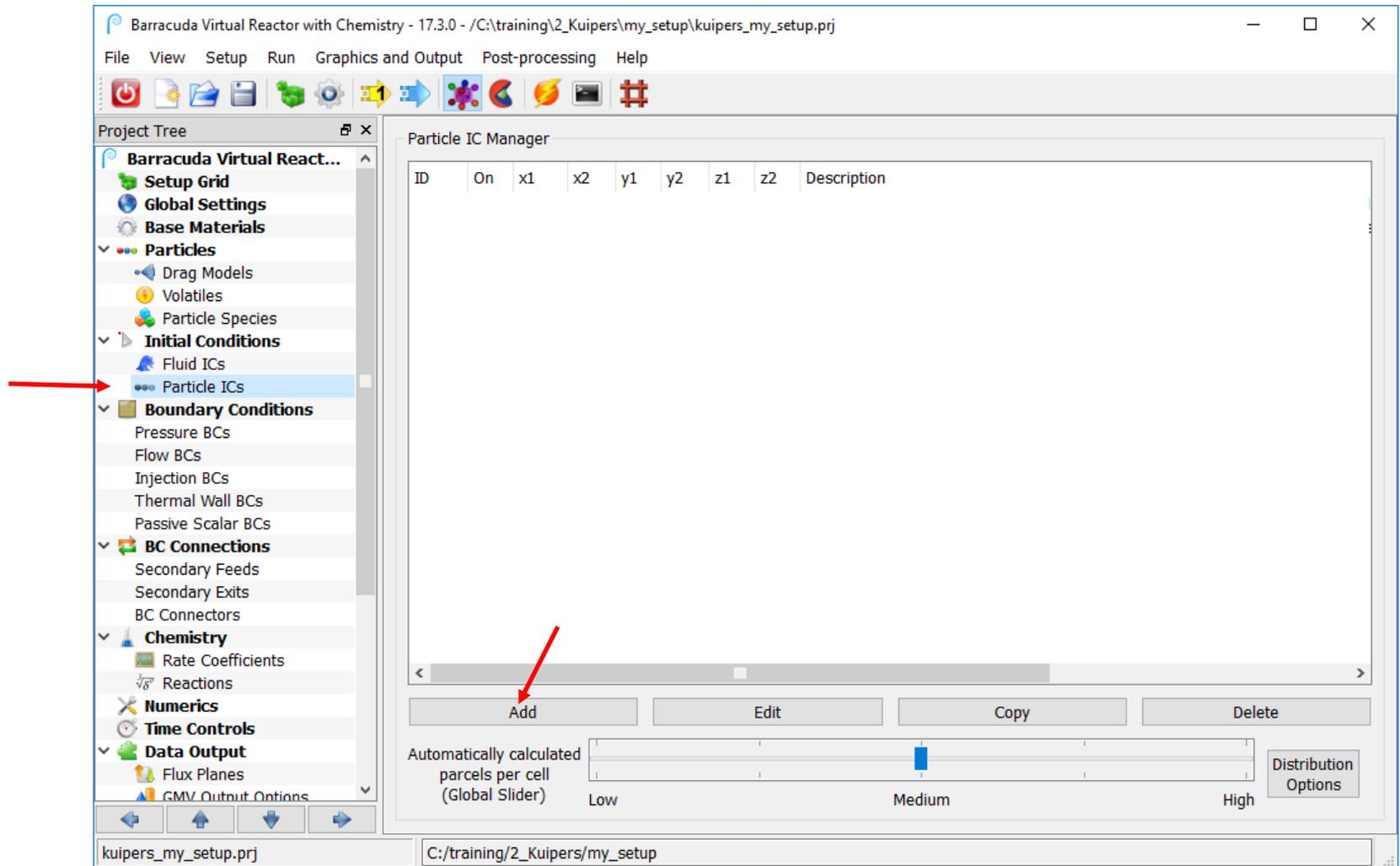
y<sub>1</sub> 0 y<sub>2</sub> 0.015

z<sub>1</sub> 0 z<sub>2</sub> 1

Comment

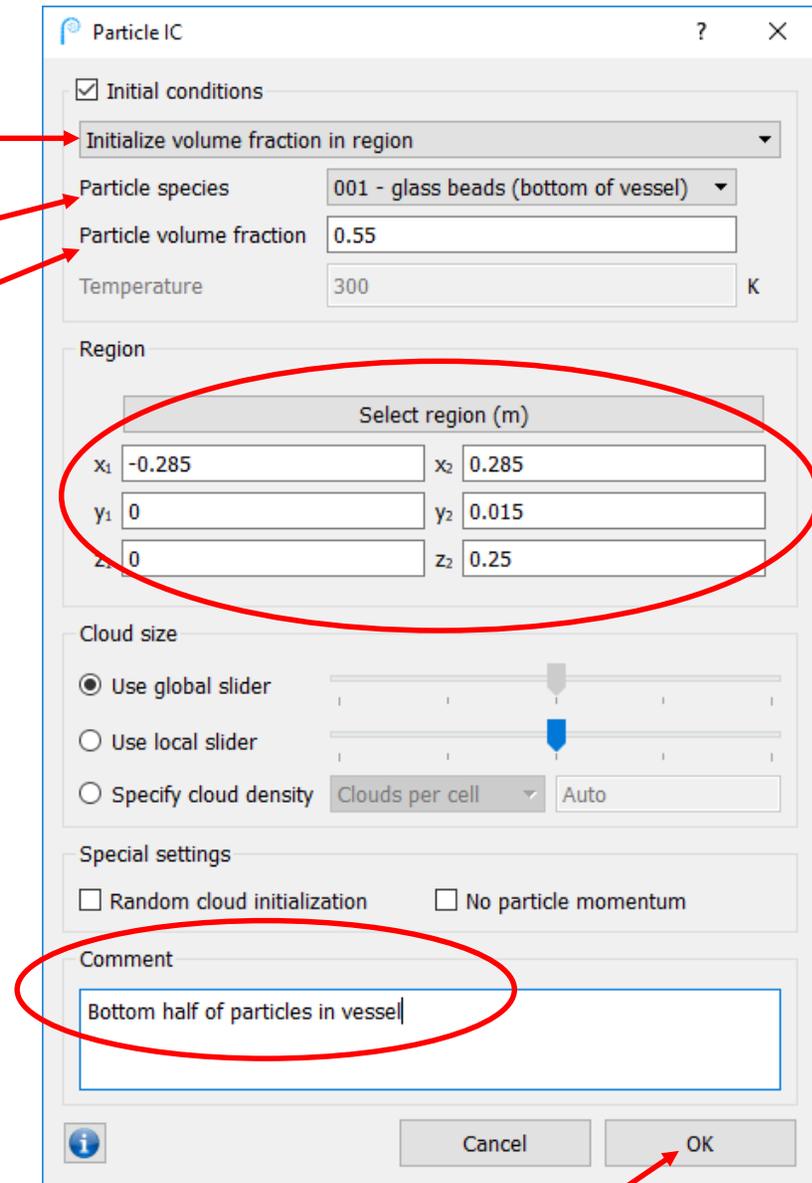
# Particle Initial Conditions

- Click on **Particle ICs**
- Click **Add**



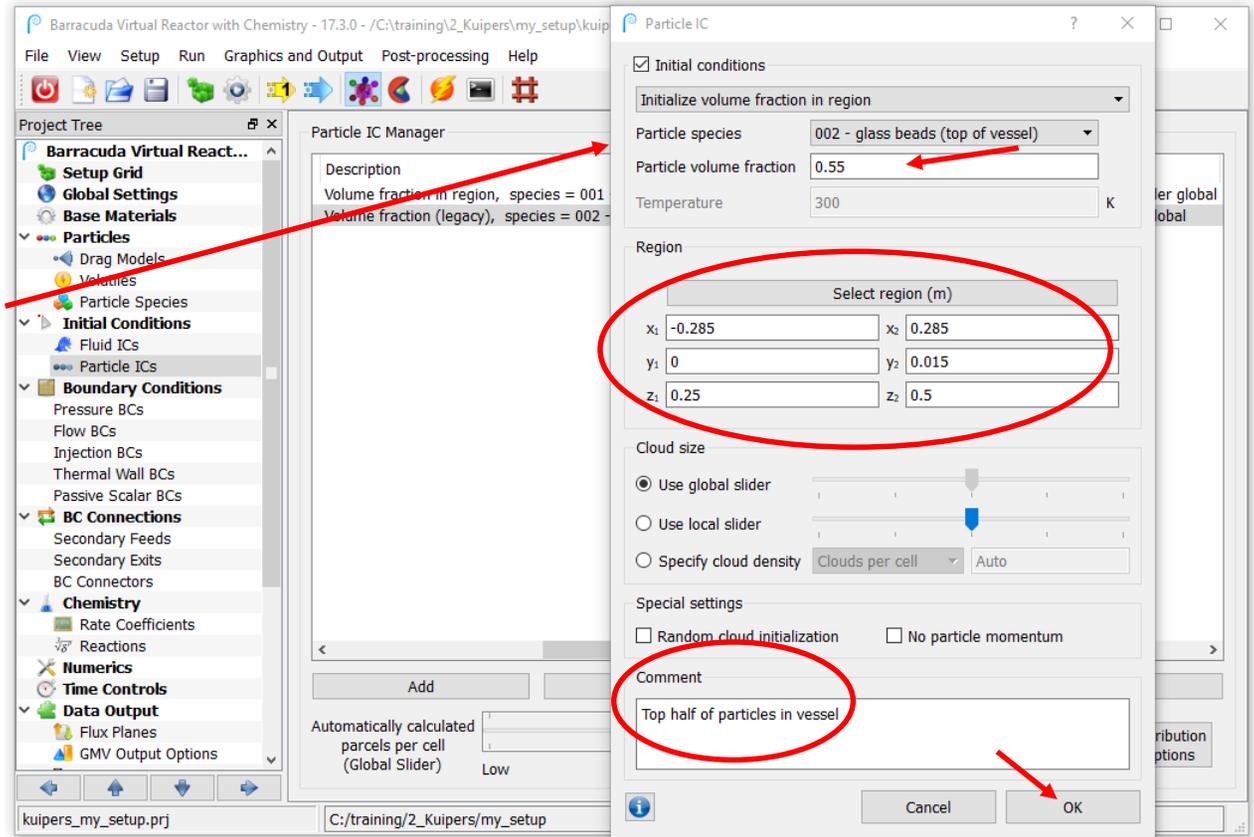
# Particle Initial Conditions

- A **Particle initialization** window will pop up
- On the Initialize tab select **Initialize volume fraction in region**
- On the **Particle species** tab select **001- glass beads (bottom of vessel)**
- Set the **Particle volume fraction** to “0.55”
- Specify the **Computational particle location** at the start of the calculation:
  - Since we want these particles to be on the bottom half of the bed, we set z1 to “[” (giving a 0) and z2 to “0.25”
- Remember to add a descriptive **Comment**
- Click **OK**



# Particle Initial Conditions

- Next, define the initial conditions for the second particle species
- Click **Add**
- On the Initialize tab select **Initialize volume fraction in region**
- On the **Particle species** tab select **002-glass beads (top of vessel)**
- Set the **Particle volume fraction** to “0.55”
- Specify the **Computational particle location**
  - Since we want these particles to be on the top half of the bed, we set z1 to “0.25” and z2 to “0.5”
- Remember to add a descriptive **Comment**
- Click **OK**



# Pressure Boundary Conditions

- Click on **Pressure BCs**
- Click **Add**

The screenshot displays the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The interface includes a menu bar (File, View, Setup, Run, Graphics and Output, Post-processing, Help) and a toolbar with various icons. On the left, the Project Tree is expanded to show "Boundary Conditions", with "Pressure BCs" selected. A red arrow points to this selection. The main area is the "Pressure BC Manager" window, which contains a table with columns: ID, Dir, x1, x2, y1, y2, z1, z2, Area, Pressure, Temp, K-Fact, Fluid properties, Particle, and Flux plane. Below the table, there are buttons for "Add", "Edit", "Copy", and "Delete". A red arrow points to the "Add" button. Below the buttons, there is a text input field for "K-factor adjustment" with the formula  $k = (1 + c_1 \theta_p) k_{\text{clean}}$  and a numeric input field for "c1" with the value "0". The status bar at the bottom shows the file name "kuipers\_my\_setup.prj" and the current directory "C:/training/2\_Kuipers/my\_setup".

# Pressure Boundary Conditions

- Define the fluid by clicking on **Define fluid species**
- The **Applied materials** window will pop up
- Click on **Add material**
- Select **AIR** and set the **Mass fraction amount** to “1”
- Click **OK**

The screenshot displays the software interface for setting pressure boundary conditions. The main window is the "Pressure BC Editor", which is divided into several sections:

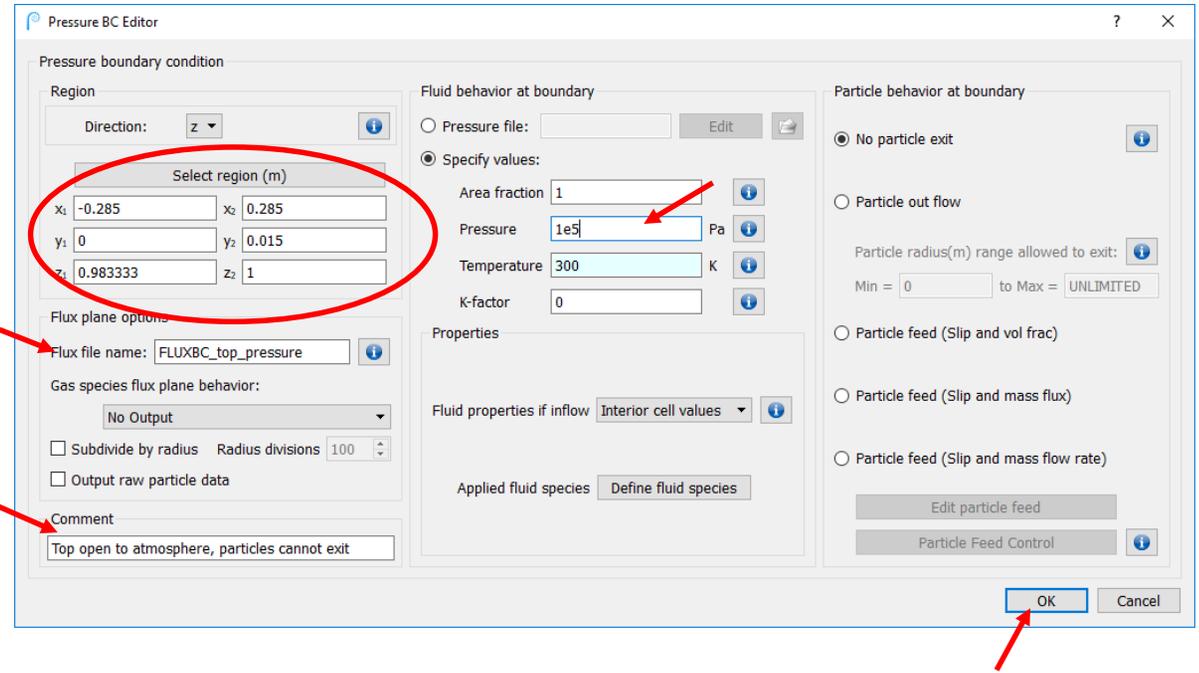
- Pressure boundary condition:** Includes a "Region" section with a "Direction" dropdown set to "z" and a "Select region (m)" button. Below this are input fields for coordinates:  $x_1$  (-0.285),  $x_2$  (0.285),  $y_1$  (0),  $y_2$  (0.015),  $z_1$  (0.983333), and  $z_2$  (1).
- Fluid behavior at boundary:** Includes radio buttons for "Pressure file" and "Specify values". Under "Specify values", there are input fields for "Area fraction" (1), "Pressure" (100000 Pa), "Temperature" (300 K), and "K-factor" (0).
- Flux plane options:** Includes a "Flux file name" field (FLUXBC\_top\_pressure) and a "Gas species flux plane behavior" dropdown set to "No Output".
- Properties:** Includes a "Fluid properties if inflow" dropdown set to "Interior cell values" and a "Define fluid species" button.

Two smaller windows are overlaid on the main interface:

- The **Applied materials** window shows a table with columns "ID", "Material", "State", and "Fraction". The "Fractions sum to:" value is 0. A red arrow points to the "Add material" button.
- The **Applied material** dialog window shows a "Project material list" table with columns "Material" and "State". The entry "AIR G" is highlighted with a red oval. Below the table, the "Fractional amount:" is set to 1. A red arrow points to this field.

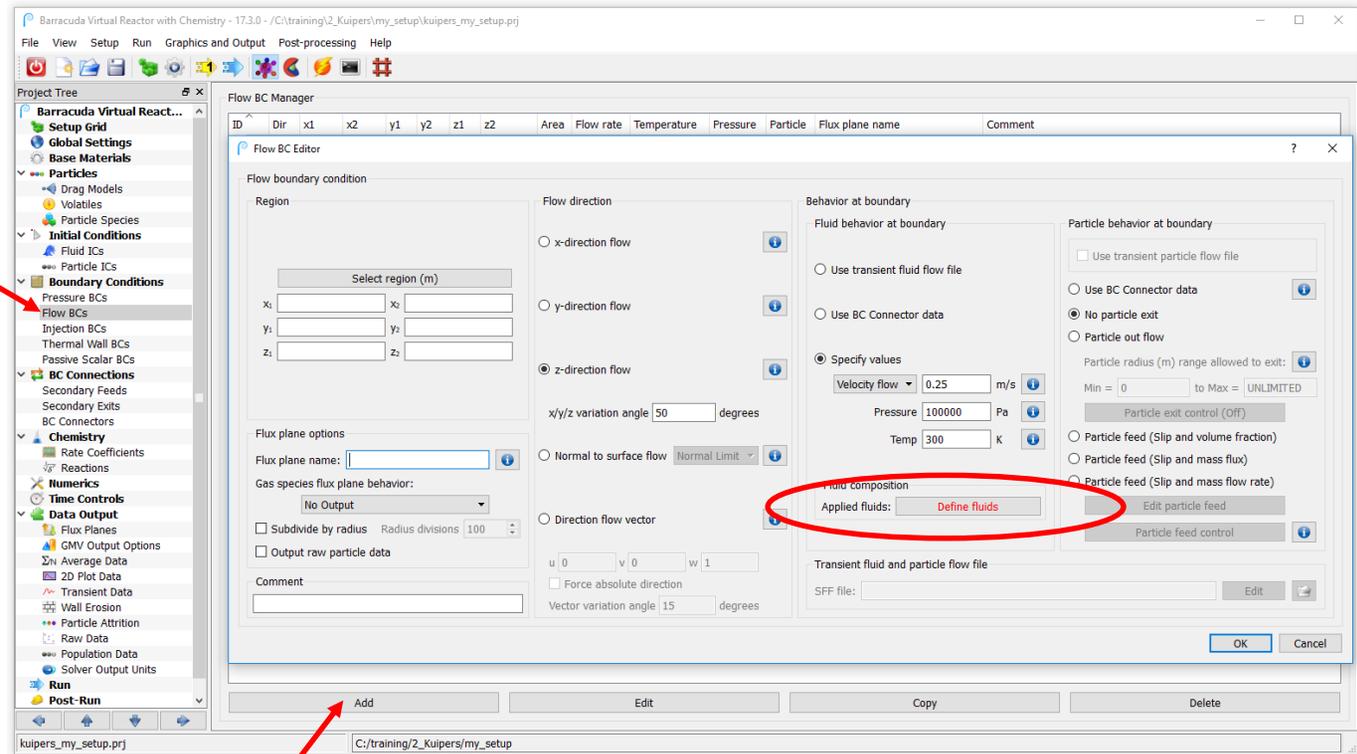
# Pressure Boundary Conditions

- Once the fluid is defined, specify the pressure
- Set the **Pressure** to “1e5” Pa
- Provide a **Flux file name** for the boundary
- Flux planes track the transient flow of fluids and solids crossing pre-defined planes and model boundaries. Use a descriptive name
  - Tip: Starting a Flux file name with “Flux\_” or “FLUX\_” will facilitate post-processing later on. This is recommended.
- Enter the plane **Location** (maximum index in z with **J**). Why these values?
- Enter a **Comment** (if desired)
- Click **OK**



# 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
- Click on **Add**
- Start by defining the fluid. Click on **Define fluids** and follow the same steps as before:
  - Click **Add material**
  - Select **AIR**
  - Set the **Mass fraction amount** to “1”
  - Click **OK**



# Flow Boundary Conditions

- Once the fluid is defined, specify the Velocity values
  - 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 up the project to bring in fluidizing air at a **Velocity Flow** of “0.25” m/s
- Set the **Pressure** to “1e5” Pa
- Leave the **Flow direction** setting at z-direction

The screenshot shows the 'Flow BC Editor' window with the following settings:

- Region:** Select region (m) with input fields for  $x_1$ ,  $x_2$ ,  $y_1$ ,  $y_2$ ,  $z_1$ , and  $z_2$ .
- Flow direction:** Radio buttons for x-direction flow, y-direction flow, and z-direction flow (selected). Includes a field for 'x/y/z variation angle' set to 50 degrees.
- Behavior at boundary:** Radio buttons for 'Fluid behavior at boundary' (Use transient fluid flow file, Use BC Connector data, Specify values - selected). Under 'Specify values', 'Velocity flow' is set to 0.25 m/s and 'Pressure' is set to 1e5 Pa. 'Temp' is set to 300 K.
- Particle behavior at boundary:** Radio buttons for 'Particle behavior at boundary' (Use BC Connector data, No particle exit - selected, Particle out flow). Includes a 'Particle radius (m) range allowed to exit' field with Min = 0 and Max = UNLIMITED.
- Flux plane options:** Flux plane name, Gas species flux plane behavior (No Output), Subdivide by radius (Radius divisions: 100), Output raw particle data.
- Fluid composition:** Applied fluids: Define fluids.
- Transient fluid and particle flow file:** SFF file field.

Red arrows in the image point to the 'z-direction flow' radio button, the 'Specify values' radio button, and the 'Velocity flow' and 'Pressure' input fields.

# Flow Boundary Conditions

- In the Kuipers setup, fluidizing air enters the system across the bottom of the bed on either side of the center jet
  - This requires that flow boundaries on either side of the jet be specified separately
  - Begin by defining the boundary for the fluidizing air entering at low indices in the x-direction
- Specify the plane **Region** as follows:
  - Set x1 to “[” and x2 to “-0.0075”
  - Set y1 to “[” and y2 to “]”
  - Set z1 to “[” and z2 to “[”
- Next, provide a **Flux plane name** for the boundary
- Enter a **Comment** (if desired)
- Click **OK**

Flow BC Editor

Flow boundary condition

Region

Select region (m)

x<sub>1</sub>: -0.285    x<sub>2</sub>: -0.0075

y<sub>1</sub>: 0    y<sub>2</sub>: 0.015

z<sub>1</sub>: 0    z<sub>2</sub>: 0.0166667

Flow direction

x-direction flow

y-direction flow

z-direction flow

x/y/z variation angle: 50 degrees

Normal to surface flow: Normal Limit

Direction flow vector

u: 0    v: 0    w: 1

Force absolute direction

Vector variation angle: 15 degrees

Flux plane options

Flux plane name: FLUXBC\_left\_fluidizing\_gas

Gas species flux plane behavior: No Output

Subdivide by radius: Radius divisions: 100

Output raw particle data

Comment: Left fluidizing gas

# Flow Boundary Conditions

- The boundary for the fluidizing air entering at high indices in the x-direction is a duplicate of the previously created boundary
- **Copy** the existing boundary condition then click on **Edit** and enter the plane **Location** as follows:
  - Set x1 to "0.0075" and x2 to "1"
  - Set y1 to "[" and y2 to "]"
  - Set z1 to "[" and z2 to "["
- Next, provide a **Flux plane name** for the boundary
- Enter a **Comment** (if desired)
- Click **OK**

The screenshot displays the Barracuda Virtual Reactor interface. The 'Flow BC Manager' window shows a table of boundary conditions:

ID	Dir	x1	x2	y1	y2	z1	z2	Area	Flow rate	Temperature	Pressure	Particle	Flux plane name
000	z	-0.285	-0.0075	0	0.015	0	0.0166667	1	0.25	300	100000	No exit	FLUXBC left fl
001	z	0.0075	0.285	0	0.015	0	0.0166667						

The 'Flow BC Editor' window is open for boundary 001. The 'Select region (m)' section is highlighted with a red circle and contains the following values:

- x1: 0.0075
- x2: 0.285
- y1: 0
- y2: 0.015
- z1: 0
- z2: 0.0166667

The 'Flux plane options' section shows:

- Flux plane name: LUXBC\_right\_fluidizing\_gas
- Gas species flux plane behavior: No Output
- Subdivide by radius:  (Radius divisions: 100)
- Output raw particle data:
- Comment: Right fluidizing gas

# Flow Boundary Conditions

- Next create a boundary for the air entering through the central jet. Enter the plane **Location** as follows:
  - Set x1 to “-0.0075” and x2 to “0.0075”
  - Set y1 to “[” and y2 to “]”
  - Set z1 to “[” and z2 to “[”
- Set **Velocity flow** to “10” m/s
- Remember to provide a **Flux plane name** for the boundary and to add a **Comment**
- Click **OK**

The screenshot shows the 'Flow BC Editor' window with the following settings:

- Region:** A red circle highlights the 'Select region (m)' section with the following values:
  - x1: -0.0075, x2: 0.0075
  - y1: 0, y2: 0.015
  - z1: 0, z2: 0.0166667
- Flow direction:** The 'z-direction flow' radio button is selected. A red arrow points to its information icon. The 'x/y/z variation angle' is set to 50 degrees.
- Behavior at boundary:** The 'Specify values' radio button is selected. The 'Velocity flow' is set to 10 m/s. The 'Pressure' is 100000 Pa and 'Temp' is 300 K.
- Flux plane options:** The 'Flux plane name' is 'LUXBC\_center\_fluidizing\_gas'. The 'Gas species flux plane behavior' is 'No Output'. There are checkboxes for 'Subdivide by radius' (checked, 100 divisions) and 'Output raw particle data'.
- Comment:** The comment field contains 'Center jet fluidizing gas'.

# 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

The screenshot shows the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The Project Tree on the left lists various simulation settings, with "Time Controls" selected. The Time Controls panel is open, showing a table for "Time step and duration settings". The first row is pre-filled with a time step of 0.0001 s and an end time of 20 s. Red arrows point to these values. Below the table is a button for "Advanced time step settings". At the bottom, the "Restart file intervals" section shows "Restart interval (IC\_###)" set to 100 simulation seconds and "Backtrack interval (IC\_)" set to 60 realtime minutes.

	Time step	End time
1.	0.0001 s	20 s
2.		
3.		
4.		
5.		

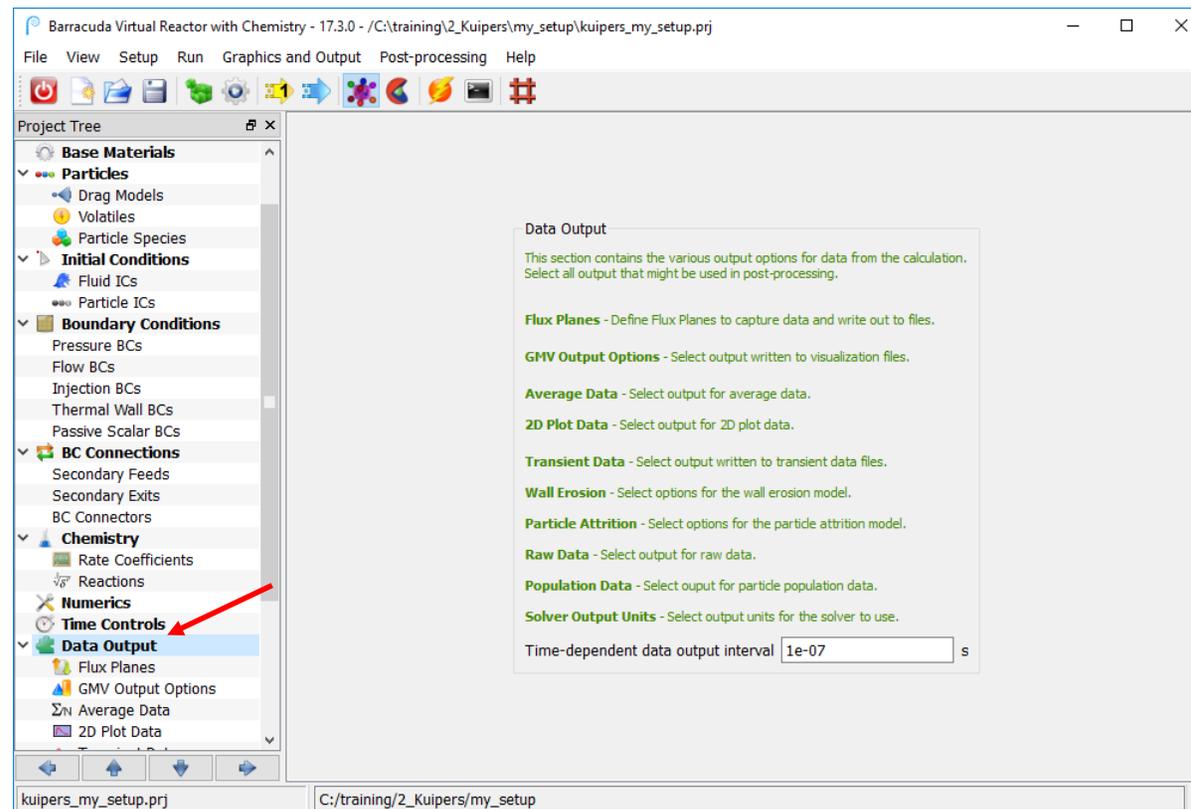
Restart file intervals

Restart interval (IC\_###) 100 simulation seconds

Backtrack interval (IC\_) 60 realtime minutes

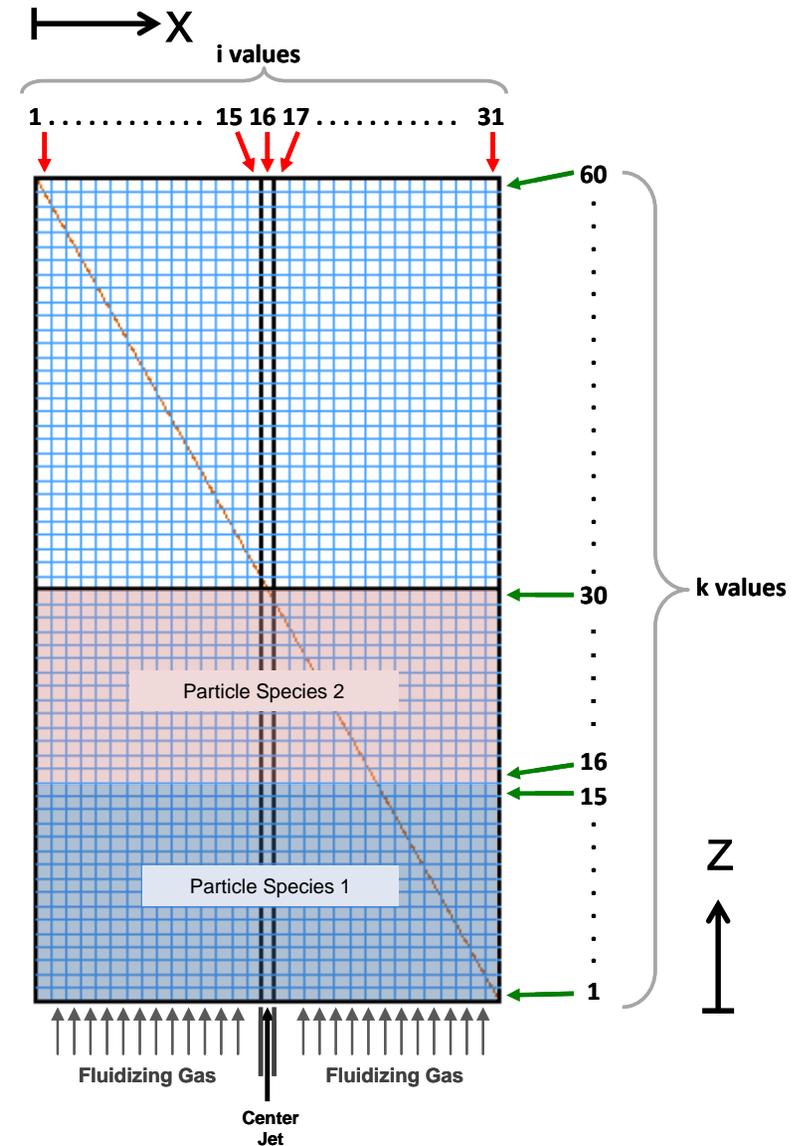
# Data Output Options

- 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
  - GMV output data – Select variables for visualization of fluid and solid states in GMV
  - Average Data – Select some Gmv output data to be averaged as the simulation runs
  - Transient Data – High frequency tracking of data at a specified location in a model



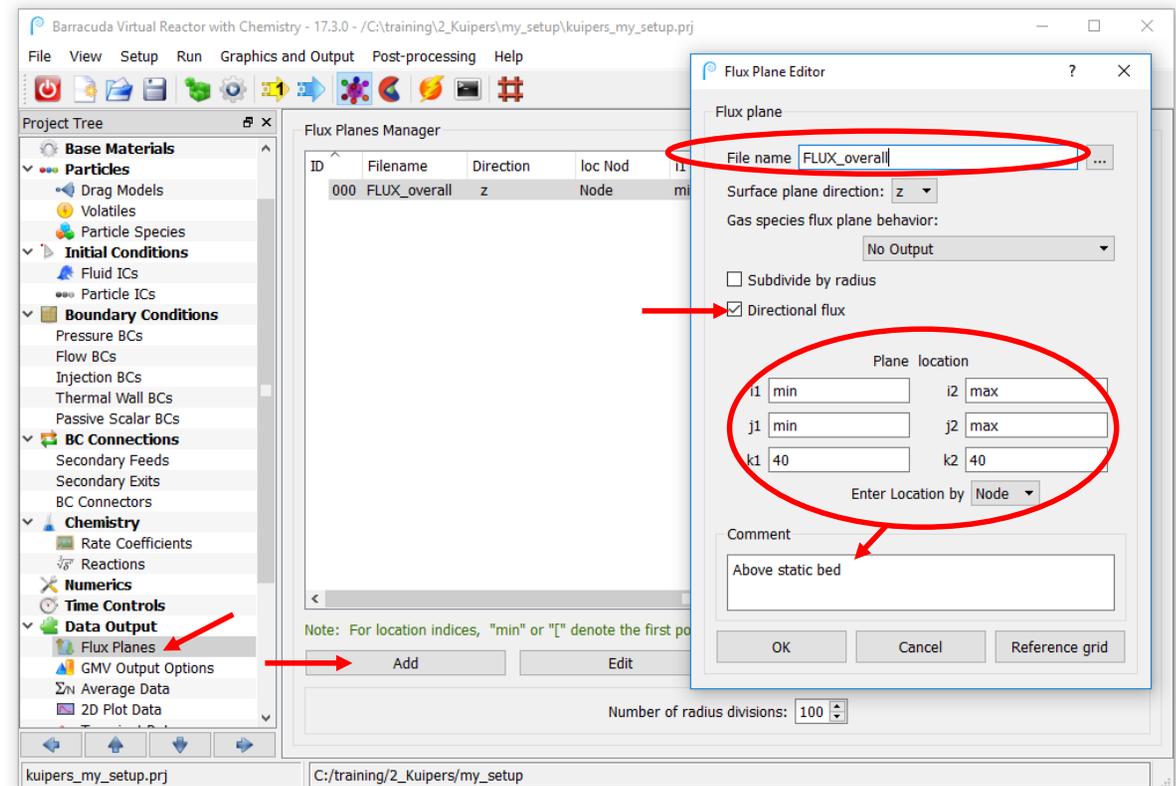
# Grid Coordinates

- 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.
- The i-, j-, and k-indices reference cells in the x-, y- and z-directions, respectively.
- Several Data Output options use either cell indices exclusively, or allow a choice between i-j-k and x-y-z:
  - Flux Planes
  - 2D Plot Data
  - Transient Data



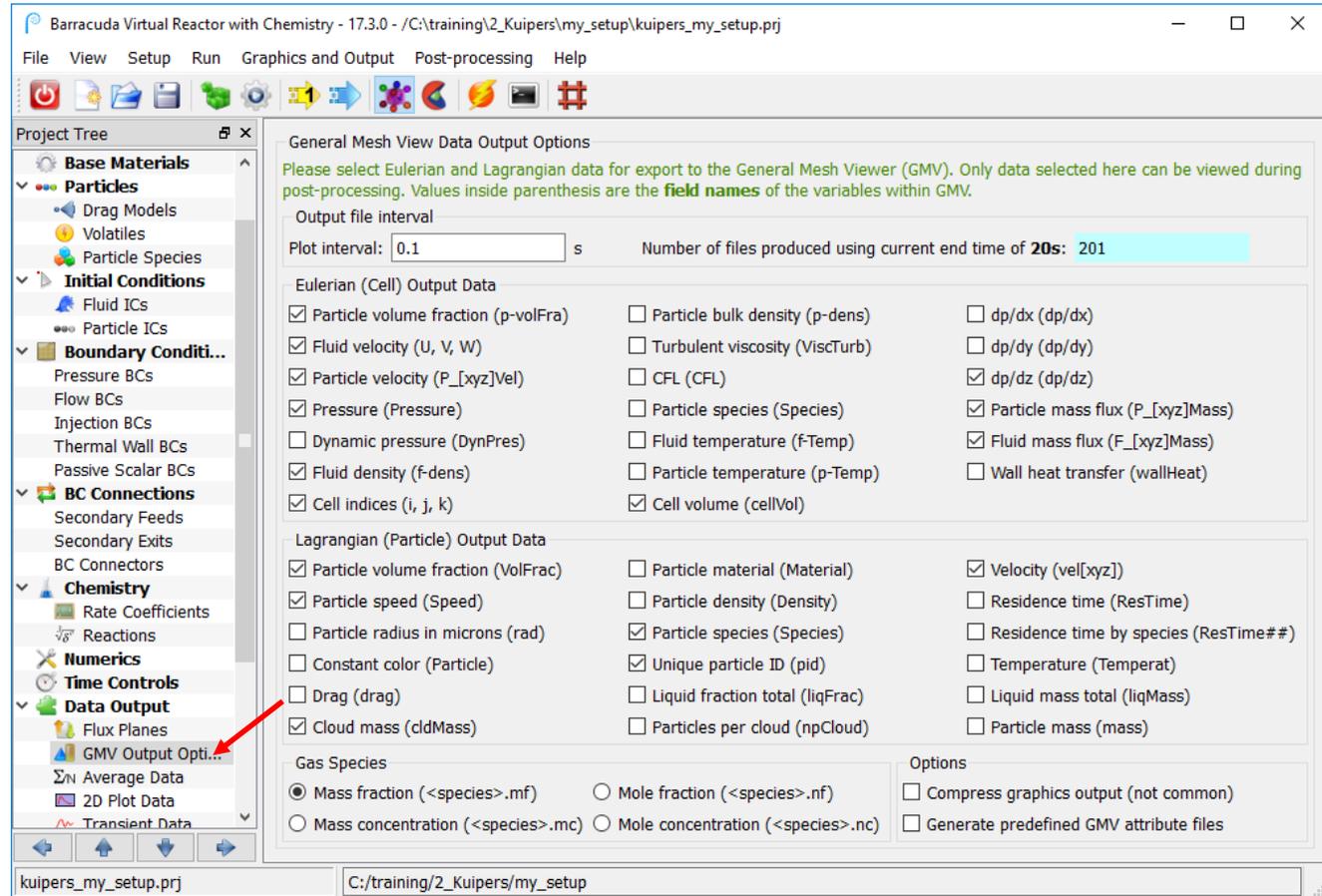
# Data Output: Flux Planes

- Click on **Flux Planes**
- Click **Add**
- Provide a descriptive **File name** for the flux plane
- Select **Directional flux**
- Enter the **Plane Location** values as shown, using i-j-k values (also called “Node”)
- Enter a **Comment** (if desired)
- Click **OK**



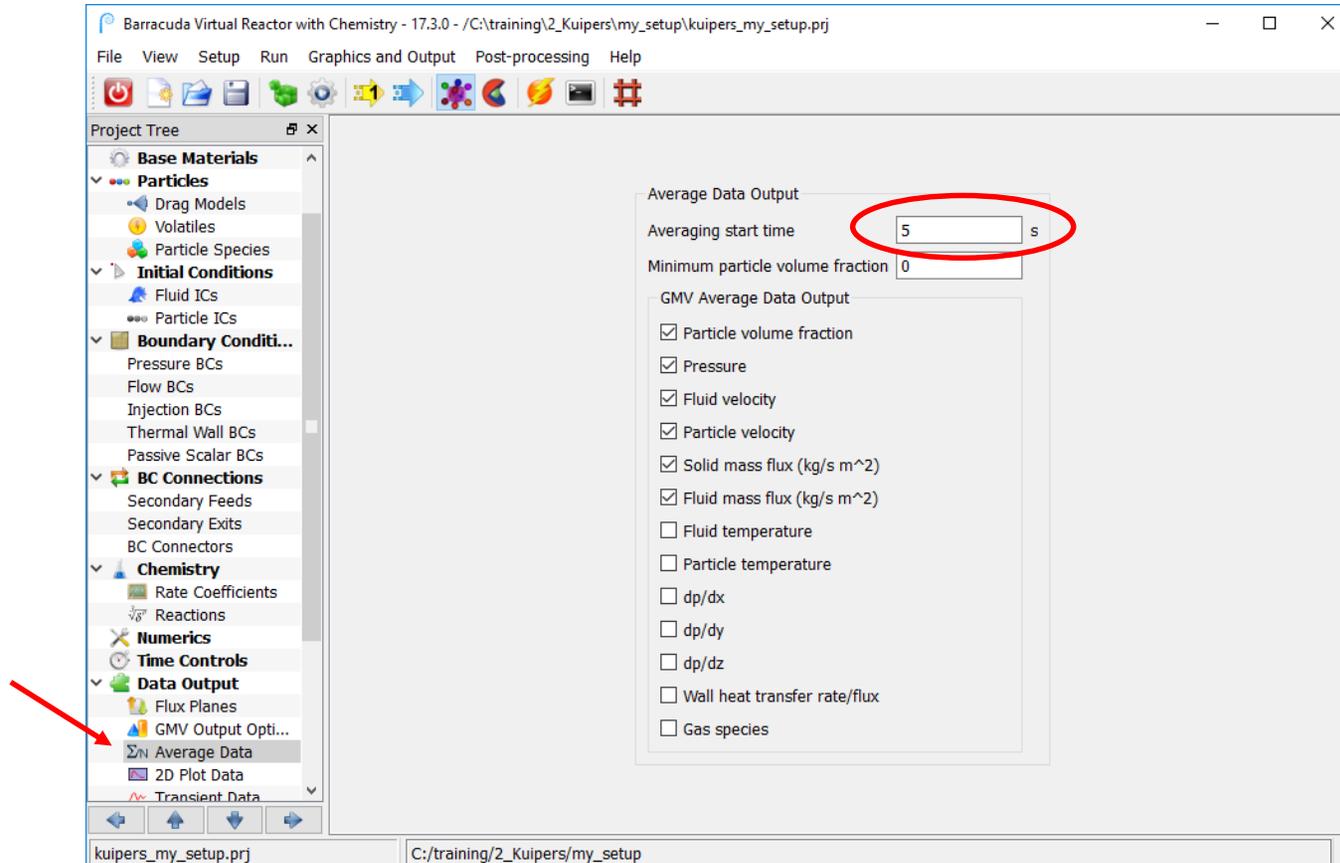
# Data Output: GMV Output Options

- Click on **GMV Output Options**
- The information selected here will be written to your Gmv output files
- The **Eulerian Output Data** is mapped to the grid
- The **Lagrangian Output Data** is mapped to particle locations
- Ensure these entities are selected since they will be required for a post-processing assignment. Show your instructor this window before proceeding



# Data Output: Average Data

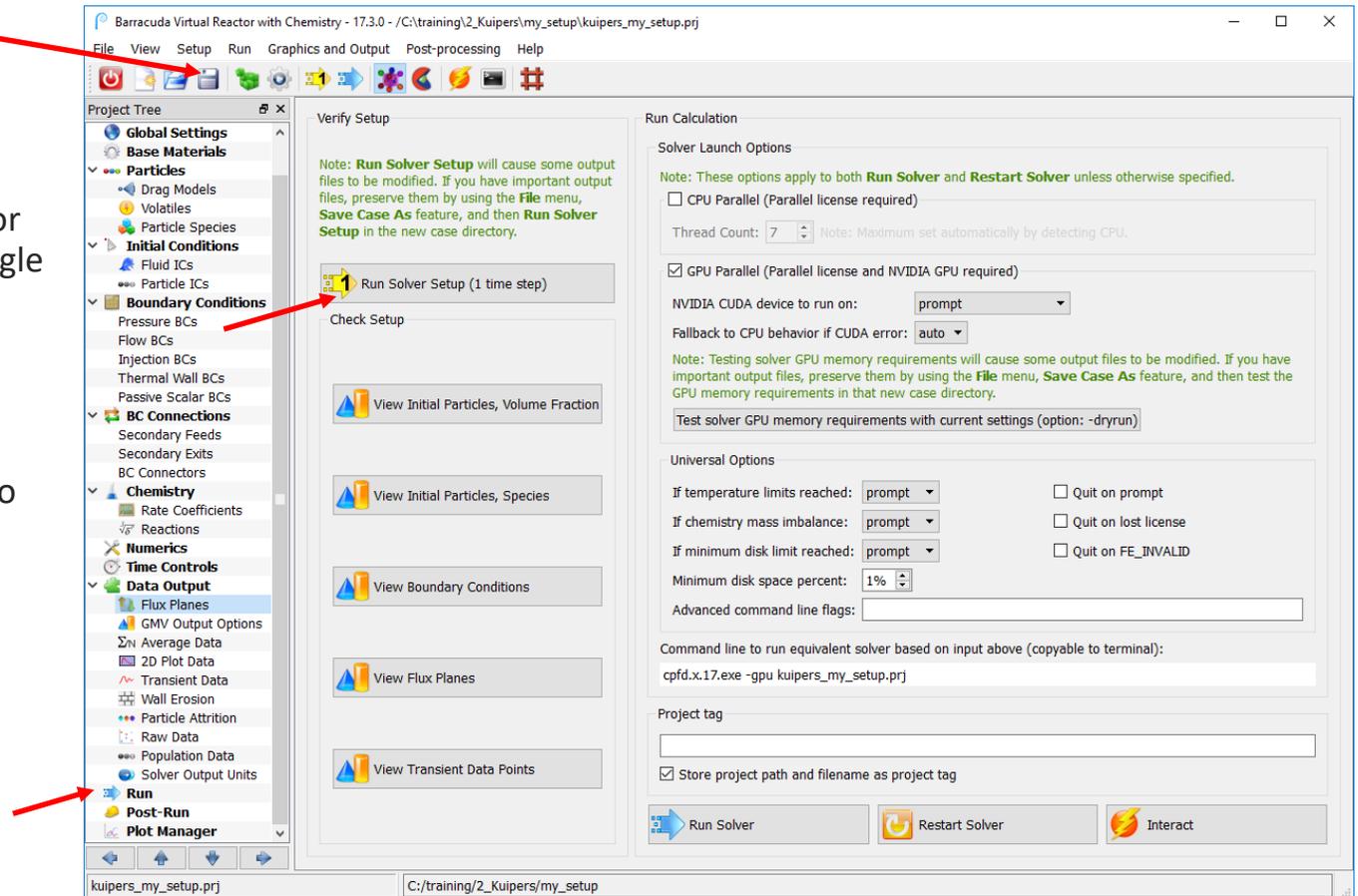
- Click on **Average Data**
- Often time-averaged data gives more insight into quasi-steady behavior than instantaneous data
- Select a **Start time for average** once you think quasi-steady behavior will have begun
- Select the data you wish to average





# Run Solver Setup

- **Save the project file**
- Click on **Run**
- Click on **Run Solver Setup**
  - This will run the simulation for one time step and write a single Gmv file
  - All boundary and initial conditions are stored in the Gmv file
  - This first Gmv file is needed to check the problem setup



# Run Solver Setup

- Solver information will be output to the run window
- Tip: 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

```
"Barracuda Virtual Reactor - 17.3.0 - /C:\training\2_Kuipers\my_setup\kuipers_my_setup.prj"
There are 1 CUDA devices.
  1 are CUDA FERMI (compute 2.0) or higher devices.
  1 are available for computation.

-----
| Barracuda Capable GPUs available for selection
-----
| Dev# | busID | Compute | Clock   | Cores | Memory Usage | Bandwidth | Compute Mode | Name
-----+-----+-----+-----+-----+-----+-----+-----+-----
|  0   |  1   |  6.1   | 1.49 GHz |  960 | 0.71 / 4.00 GB | 3.7 GB/s | DEFAULT     | GeForce GTX 1050
-----+-----+-----+-----+-----+-----+-----+-----+-----

Enter Dev#:0
Acquiring device #0 in compute mode:DEFAULT.
Waiting for access to device #0...done.
Waiting for memory on device #0...done.
Using device #0 for calculation.
Name:           GeForce GTX 1050
Compute version: 6.1
Global memory:  4096 MB
Clock rate:     1.49 GHz
CUDA cores:    960
HstToDev Bandwidth: 3615 MB
Reprise environment variable: cpfd_LICENSE=27015@127.0.0.1

Reprise Project tag: path:C:\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+00 000 0.00e+00 0.00 96 0 0 0
Dumping Gmv.00000
1.00000e-04 1.000e-04 002 2.07e-08 001 0.00e+00 000 0.00e+00 002 1.19e-12 26 8.59e-07 0.04 95 0 0 0
Thu Jan 04 14:41:28 2018
C:\training\2_Kuipers\my_setup>
```

# Checking your Setup

- Before letting your calculation run, check the following:
  - are your particles defined correctly and located where you want them?
  - are your boundary conditions in the right place?
  - are your transient data points located where you want them?

The screenshot shows the Barracuda Virtual Reactor software interface. The 'Verify Setup' panel is active, displaying a note: "Note: Run Solver Setup will cause some output files to be modified. If you have important output files, preserve them by using the File menu, Save Case As feature, and then Run Solver Setup in the new case directory." Below this note is a button labeled "Run Solver Setup (1 time step)".

The 'Check Setup' section is circled in red and contains five buttons with warning icons:

- View Initial Particles, Volume Fraction
- View Initial Particles, Species
- View Boundary Conditions
- View Flux Planes
- View Transient Data Points

The 'Run Calculation' panel is also visible, showing 'Solver Launch Options' with checkboxes for 'CPU Parallel' and 'GPU Parallel'. The 'GPU Parallel' option is checked. Below this are 'Universal Options' for temperature, chemistry mass imbalance, and minimum disk limit, each with a dropdown menu and a 'Quit on...' checkbox. At the bottom, there are buttons for 'Run Solver', 'Restart Solver', and 'Interact'.

# Checking your Setup - Particles

- Click on **View Initial Particles, Volume Fraction**
- Verify the initial location of the particles in the bed
- To change the view:
  - Use the mouse buttons: left button rotates, center button translates, and right button zooms OR
  - Use the sliders at the top in the GUI

The screenshot displays the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The interface includes a Project Tree on the left, a central workspace, and a Run Calculation panel on the right. A red arrow points to the "View Initial Particles, Volume Fraction" button in the central workspace. An inset window titled "GMV V4.6.1 built 2015-03-18" is overlaid on the main window, showing a dialog box with a red circle around the "Current file" field and coordinate values: Twist: 0,0; Elev.: -15,0; Azim.: -40,0. The main workspace shows a 3D visualization of the reactor bed with a color scale for "Particles VolFrac" ranging from 0.3266944 (blue) to 0.5486134 (red). The Run Calculation panel shows "Solver Launch Options" with "GPU Parallel" checked and "NVIDIA CUDA device to run on:" set to "prompt".

# Checking your Setup – Particle Species

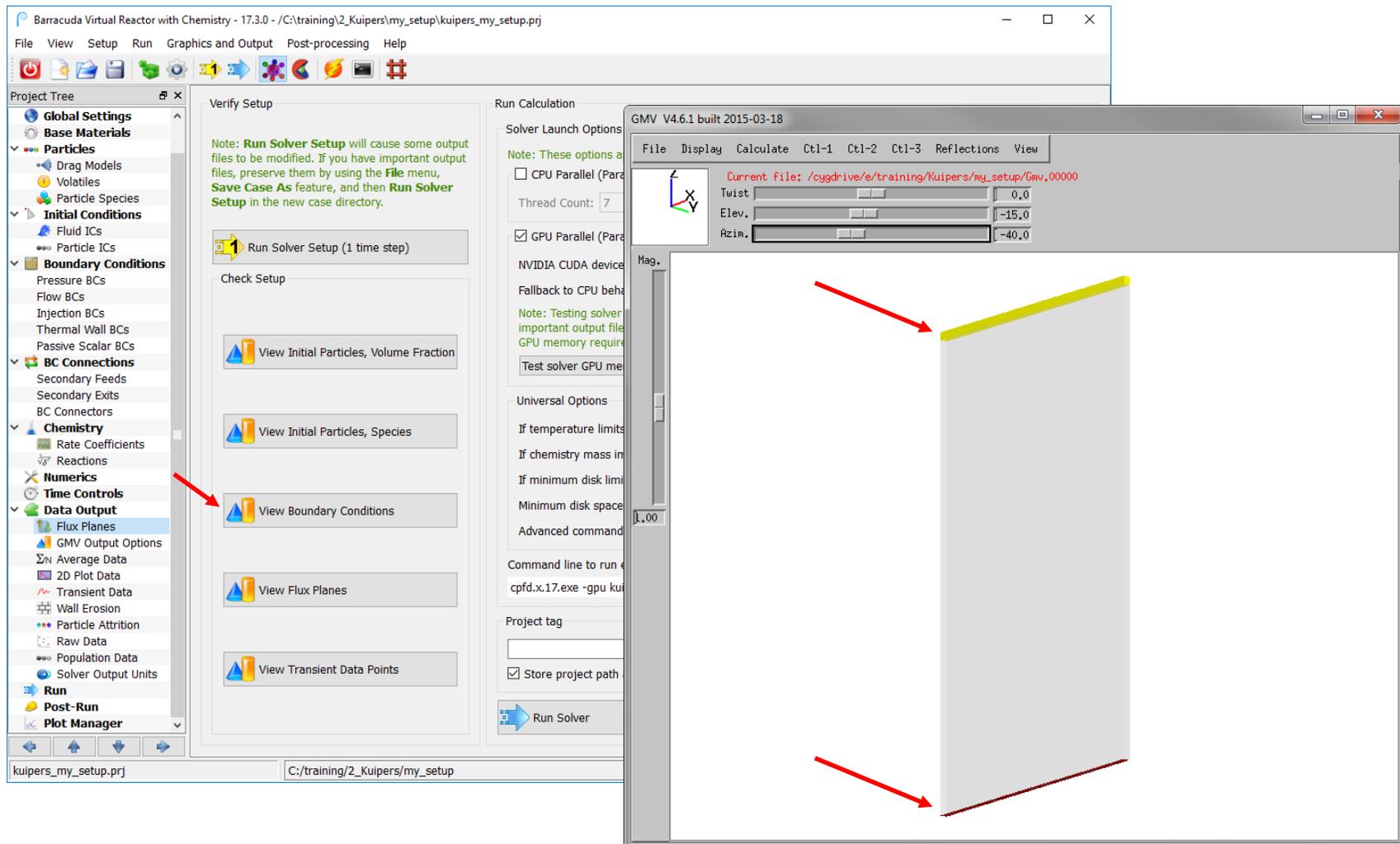
- Click on **View Initial Particles, Species**
- Verify the initial location of the two particle species in the bed

The screenshot displays the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The interface is divided into several panels:

- Project Tree:** A hierarchical list of settings including Global Settings, Base Materials, Particles, Initial Conditions, Boundary Conditions, BC Connections, Chemistry, Numerics, Time Controls, Data Output, Run, Post-Run, and Plot Manager. A red arrow points to the "View Initial Particles, Species" option in the "Verify Setup" panel.
- Verify Setup:** A central panel with a note: "Note: Run Solver Setup will cause some output files to be modified. If you have important output files, preserve them by using the File menu, Save Case As feature, and then Run Solver Setup in the new case directory." Below the note are several buttons: "Run Solver Setup (1 time step)", "Check Setup", "View Initial Particles, Volume Fraction", "View Initial Particles, Species", "View Boundary Conditions", "View Flux Planes", and "View Transient Data Points".
- Run Calculation:** A panel on the right with various solver options, including "Solver Launch Option", "CPU Parallel", "GPU Parallel", and "NVIDIA CUDA device".
- GMV V4.6.1 built 2015-03-18:** A separate window showing a 3D plot of "Particles Species". The plot displays a rectangular domain with a color gradient from blue (1.0) to red (2.0). The plot is titled "Particles Species" and has a value of "0.0000000e+00" displayed in the top right corner. The plot shows a red layer on top and a blue layer on the bottom, representing the initial location of the two particle species in the bed.

# Checking your Setup – Boundary Conditions

- Click on **View Boundary Conditions**
- Verify that the boundary conditions are in the right place



# Checking your Setup – Flux Plane Location

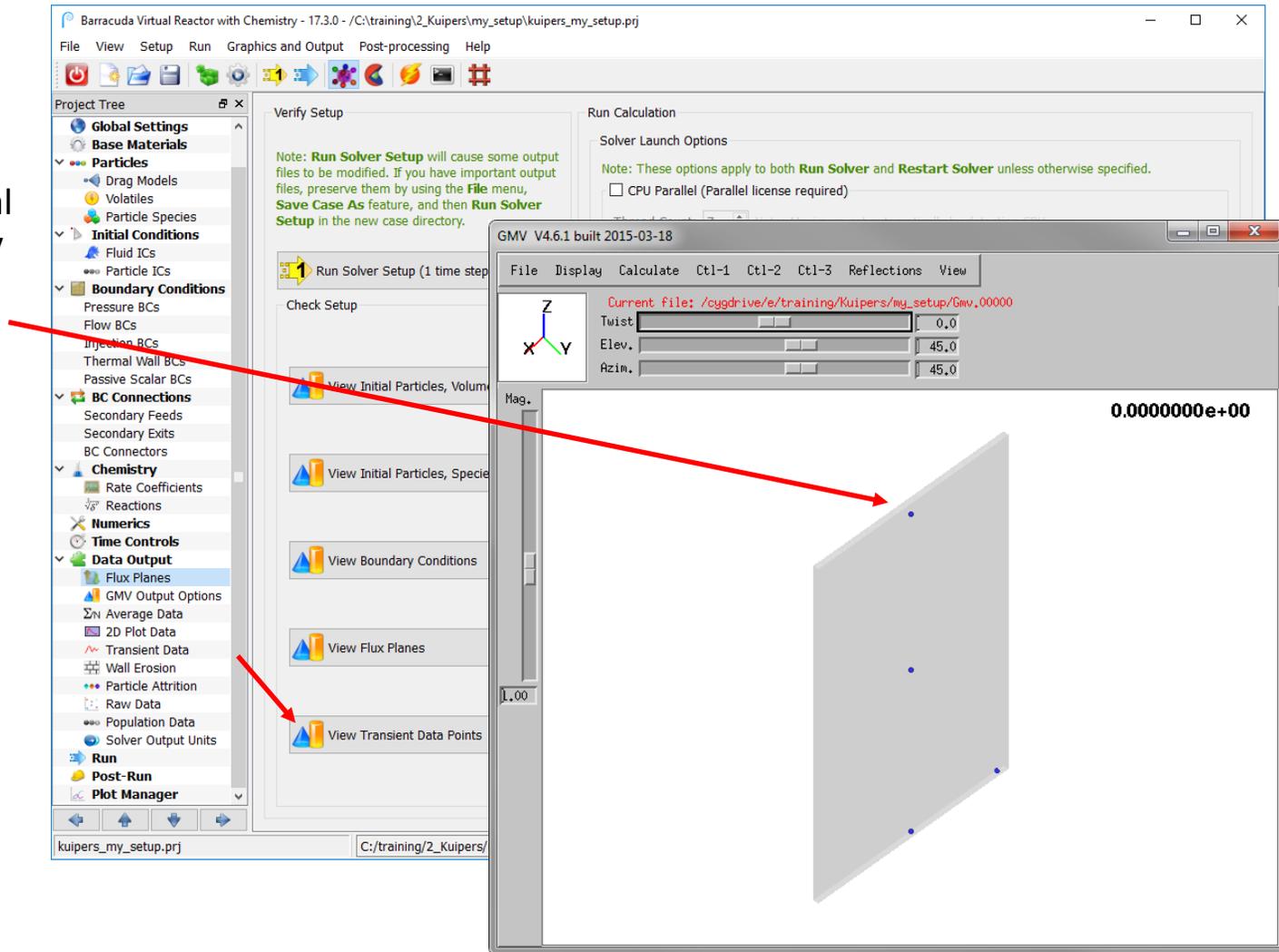
- Click on **View Flux Planes**
- Verify that the flux planes are in the right place

The screenshot displays the Barracuda Virtual Reactor software interface. The main window is titled "Barracuda Virtual Reactor with Chemistry - 17.3.0 - /C:/training/2\_Kuipers/my\_setup/kuipers\_my\_setup.prj". The interface is divided into several panels:

- Project Tree:** Located on the left, it shows a hierarchical view of the simulation setup. The "Data Output" section is expanded, and "Flux Planes" is selected. A red arrow points from "Flux Planes" in the Project Tree to the "View Flux Planes" button in the Verify Setup panel.
- Verify Setup:** This panel contains a note: "Note: Run Solver Setup will cause some output files to be modified. If you have important output files, preserve them by using the File menu, Save Case As feature, and then Run Solver Setup in the new case directory." Below the note are several buttons: "Run Solver Setup (1 time step)", "Check Setup", "View Initial Particles, Volume Fraction", "View Initial Particles, Species", "View Boundary Conditions", "View Flux Planes", and "View Transient Data Points".
- Run Calculation:** This panel contains "Solver Launch Options" and "Universal Options". The "GPU Parallel" option is checked. A "Run Solver" button is at the bottom.
- GMV V4.6.1 built 2015-03-18:** This is a separate window showing a 3D visualization of the reactor. The current file is "/cygdrive/e/training/Kuipers/my\_setup/Gmv.00000". The visualization shows a rectangular reactor vessel with a horizontal purple line representing a flux plane. A red arrow points from the "View Flux Planes" button in the Verify Setup panel to this purple line. The magnitude of the flux is displayed as "0.0000000e+00".

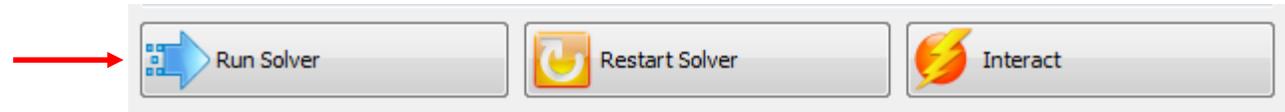
# Checking your Setup – Transient Data Point Location

- Click on **View Transient Data Points**
- Verify that the transient data points are in the right place
- Show your instructor your initial particle locations and boundary conditions before proceeding



# Executing the Simulation

- To begin running your calculation, click on **Run Solver**



- Examine the solver window

```

C:\> Select "Barracuda Virtual Reactor - 17.3.0 - /C:\training\2_Kuipers\my_setup\kuipers_my_setup.prj" - cpfd.x.17.exe -gpu kuipers_my_setup.prj

Compute version: 6.1
Global memory: 4096 MB
Clock rate: 1.49 GHz
CUDA cores: 960
HstToDev Bandwidth: 3844 MB
Reprise environment variable: cpfd_LICENSE=27015@127.0.0.1

Reprise Project tag: path:C:\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 96 0 0 0
Dumping Gmv.00000
1.00000e-04 1.000e-04 002 2.07e-08 001 0.00e+00 000 0.00e+00 002 1.19e-12 26 8.59e-07 0.04 95 0 0 0
2.00000e-04 1.000e-04 002 3.76e-08 003 2.06e-09 000 0.00e+00 003 3.89e-09 22 7.00e-07 0.04 95 0 0 0
3.00000e-04 1.000e-04 002 6.99e-08 003 1.81e-09 000 0.00e+00 003 2.97e-09 19 6.95e-07 0.04 95 0 0 0
4.00000e-04 1.000e-04 002 8.21e-08 003 9.70e-10 000 0.00e+00 003 1.32e-09 18 5.78e-07 0.04 96 0 0 0
5.00000e-04 1.000e-04 003 7.16e-08 003 6.11e-10 000 0.00e+00 003 7.26e-10 18 4.77e-07 0.04 97 0 0 0
6.00000e-04 1.000e-04 003 6.49e-08 003 4.17e-10 000 0.00e+00 003 4.71e-10 18 3.96e-07 0.04 98 0 0 0
7.00000e-04 1.000e-04 003 7.96e-08 003 2.78e-10 000 0.00e+00 003 3.16e-10 17 5.63e-07 0.04 97 0 0 0
8.00000e-04 1.000e-04 003 9.95e-08 002 6.74e-08 000 0.00e+00 003 2.12e-10 16 7.92e-07 0.04 95 0 0 0
    
```

# Solver Output Window

**Current Simulation Time**

**Current Time step**

**Solver Convergence Data**

**CFL number:**  
Typically safe to run between 0.7 – 1.5

```

C:\> Select "Barracuda Virtual Reactor - 17.3.0 - /C:\training\2_Kuipers\my_setup\kuipers_my_setup.prj" - cpfd.x.17.exe -gpu kuipers_my_setup.prj

Global memory:      4096 MB
Clock rate:         1.49 GHz
CUDA cores:         960
HstToDev Bandwidth: 3737 MB
Reprise environment variable: cpfd_LICENSE=27015@127.0.0.1

Reprise Project tag: path:C:\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 96 0 0 0
Dumping Gmv.00000
1.00000e-04 1.000e-04 002 2.07e-08 001 0.00e+00 000 0.00e+00 002 1.19e-12 26 8.59e-07 0.04 95 0 0 0
2.00000e-04 1.000e-04 002 3.76e-08 003 2.06e-09 000 0.00e+00 003 3.89e-09 22 7.00e-07 0.04 95 0 0 0
3.00000e-04 1.000e-04 002 6.99e-08 003 1.81e-09 000 0.00e+00 003 2.97e-09 19 6.95e-07 0.04 95 0 0 0
4.00000e-04 1.000e-04 002 8.21e-08 003 9.70e-10 000 0.00e+00 003 1.32e-09 18 5.78e-07 0.04 96 0 0 0
  
```

If CFL number is significantly below 0.7: 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**
- Only enter information you wish to change
- Here, we can increase the **Time step** to “1e-3” seconds. Enter the new time and click **Submit**

The screenshot shows the Barracuda solver interface. At the top, there are three buttons: "Run Solver", "Restart Solver", and "Interact". A red arrow points to the "Interact" button. Below this is the "Interact Barracuda" dialog box. The dialog box has a title bar "Interact Barracuda" and a close button. It contains several sections:

- Time control:** "Time step" is set to 0.001 (circled in red), "End time" is empty.
- CFL:** "Min CFL" and "Max CFL" are empty.
- QFL:** "Min QFL" and "Max QFL" are empty.
- Dump output:** Four checkboxes are present: "Dump restart (IC\_###)", "Dump graphics (SnapGmv file)", "Dump population (POPUL...)", and "Dump all particles (p\_all...snap)".
- Reread Files:** Two checkboxes are present: "Reread BC input files" and "Reread PSD input files".
- Output frequency:** "Print interval (Terminal)", "Plot interval (GMV)", "Restart interval (IC\_###)", and "Backtrack interval (IC\_)" are empty. There is a checkbox "Start new plot intervals now" and a note "Default is after the next regular plot."
- Residuals:** A table with columns "Iteration" and "Residual". Rows are "Volume", "Pressure", and "Velocity".
- Misc:** Three checkboxes: "Reset wear to 0", "Reset attrition to 0", and "Reset particle mass deficit at BCs to 0".

At the bottom of the dialog box, there are three buttons: "Submit" (with a green checkmark), "Clear" (with a red X), and "Exit" (with a red X). A red arrow points to the "Submit" button. In the top right corner of the dialog box, it says "Version: 3.51" and "Build Date: 2017-08-24". The "cpfd" logo and "COMPUTATIONAL PARTICLE FLUID DYNAMICS" text are also visible.

# Dynamic Solver Interaction

- Notice the solver begins running at the new time step
- Tip: It is Generally recommended to slowly raise the time step. It is also advisable to request a restart file before changing calculation parameters

```

CAL Select "Barracuda Virtual Reactor - 17.3.0 - /C:\training\2_Kuipers\my_setup\kuipers_my_setup.prj" - cpfd.x.17.exe -gpu kuipers_my_setup.prj
1.97000e-02  1.000e-04  002  9.18e-08  002  3.54e-09  000  0.00e+00  002  3.06e-09  10  3.36e-07  0.03  21  0  0  0
-----
      t      dt  Vol  Vol  u    u    v    v    w    w    p    p    CFL  Low Med  Hi  R
      s      s  itr  err  itr  err  itr  err  itr  err  itr  err
-----
1.98000e-02  1.000e-04  002  9.20e-08  002  3.44e-09  000  0.00e+00  002  3.02e-09  10  3.03e-07  0.03  21  0  0  0
1.99000e-02  1.000e-04  002  9.40e-08  002  3.92e-09  000  0.00e+00  002  2.87e-09  10  3.01e-07  0.03  21  0  0  0
2.00000e-02  1.000e-04  002  9.51e-08  002  3.14e-09  000  0.00e+00  002  2.88e-09  10  3.28e-07  0.03  21  0  0  0
2.01000e-02  1.000e-04  002  9.83e-08  002  3.55e-09  000  0.00e+00  002  3.03e-09  10  3.99e-07  0.03  21  0  0  0
2.02000e-02  1.000e-04  002  9.89e-08  002  3.86e-09  000  0.00e+00  002  2.98e-09  10  3.65e-07  0.03  21  0  0  0
Remote set dt=0.001000s
2.12000e-02  1.000e-03  006  9.93e-08  002  8.65e-09  000  0.00e+00  002  1.00e-08  83  6.44e-07  0.31  21  0  0  0
2.22000e-02  1.000e-03  003  8.70e-08  002  3.82e-08  000  0.00e+00  002  2.12e-08  78  3.27e-07  0.31  21  0  0  0
2.32000e-02  1.000e-03  006  9.38e-08  002  5.64e-08  000  0.00e+00  002  2.59e-08  82  3.69e-07  0.31  21  0  0  0
2.42000e-02  1.000e-03  004  9.94e-08  002  4.86e-08  000  0.00e+00  002  3.13e-08  78  3.75e-07  0.31  21  0  0  0
2.52000e-02  1.000e-03  011  1.51e-08  002  5.12e-08  000  0.00e+00  002  3.23e-08  117  9.54e-08  0.31  21  0  0  0
2.62000e-02  1.000e-03  001  6.27e-08  002  5.27e-08  000  0.00e+00  002  7.24e-08  82  8.61e-08  0.31  21  0  0  0
-----
      t      dt  Vol  Vol  u    u    v    v    w    w    p    p    CFL  Low Med  Hi  R
      s      s  itr  err  itr  err  itr  err  itr  err  itr  err
-----
  
```

# Let it Run

- If everything is as expected, let the solver run
- Next we'll work through a post-processing assignment