

# CFB Erosion: Setup and Analysis

CPFD Software

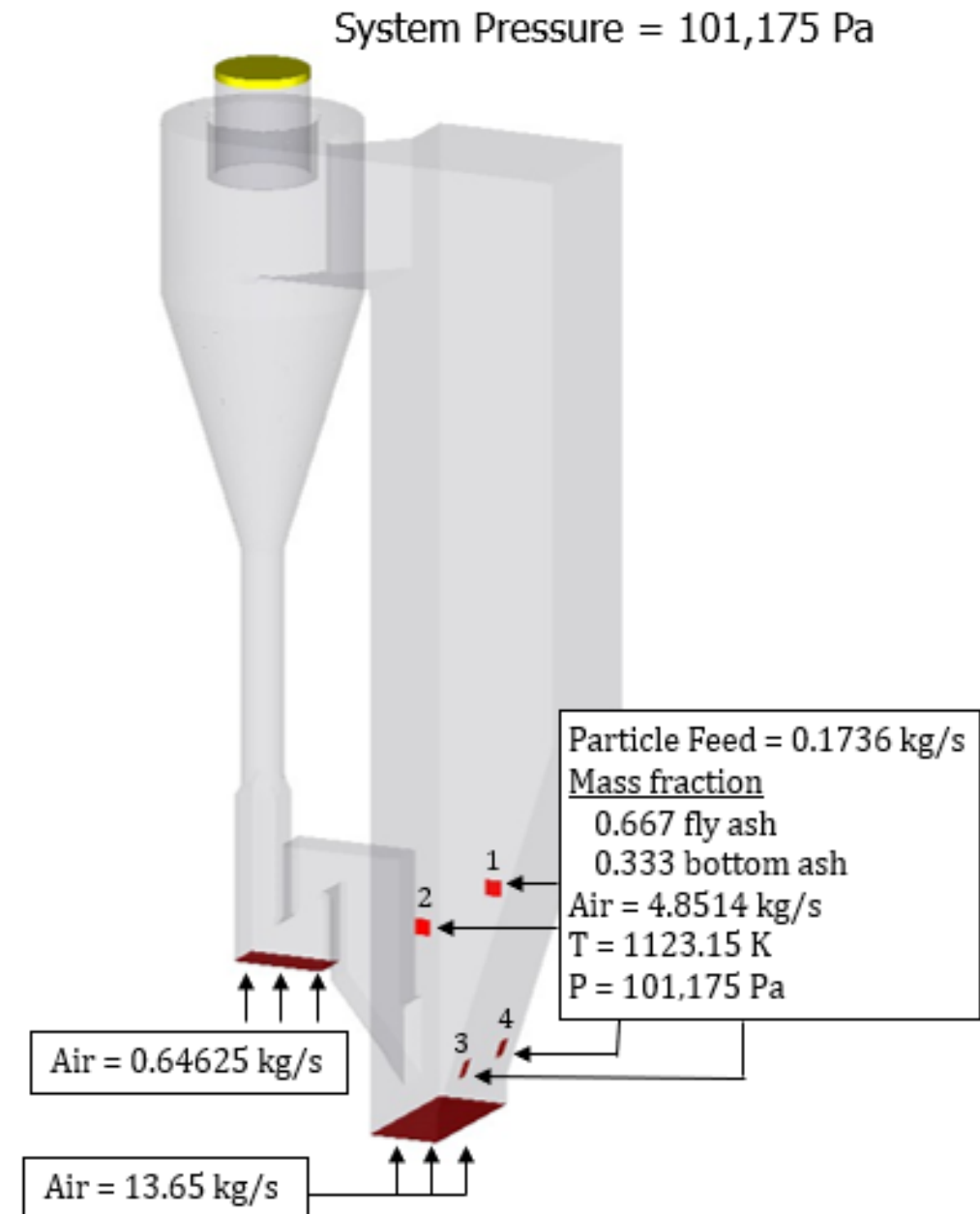
[www.cpfd-software.com](http://www.cpfd-software.com)

# Circulating Fluidized Bed (CFB)

Model of circulating fluidized bed (CFB) based on the system operating at the 40 MW Strongoli power plant in Italy.

Start with Case 1: Baseline

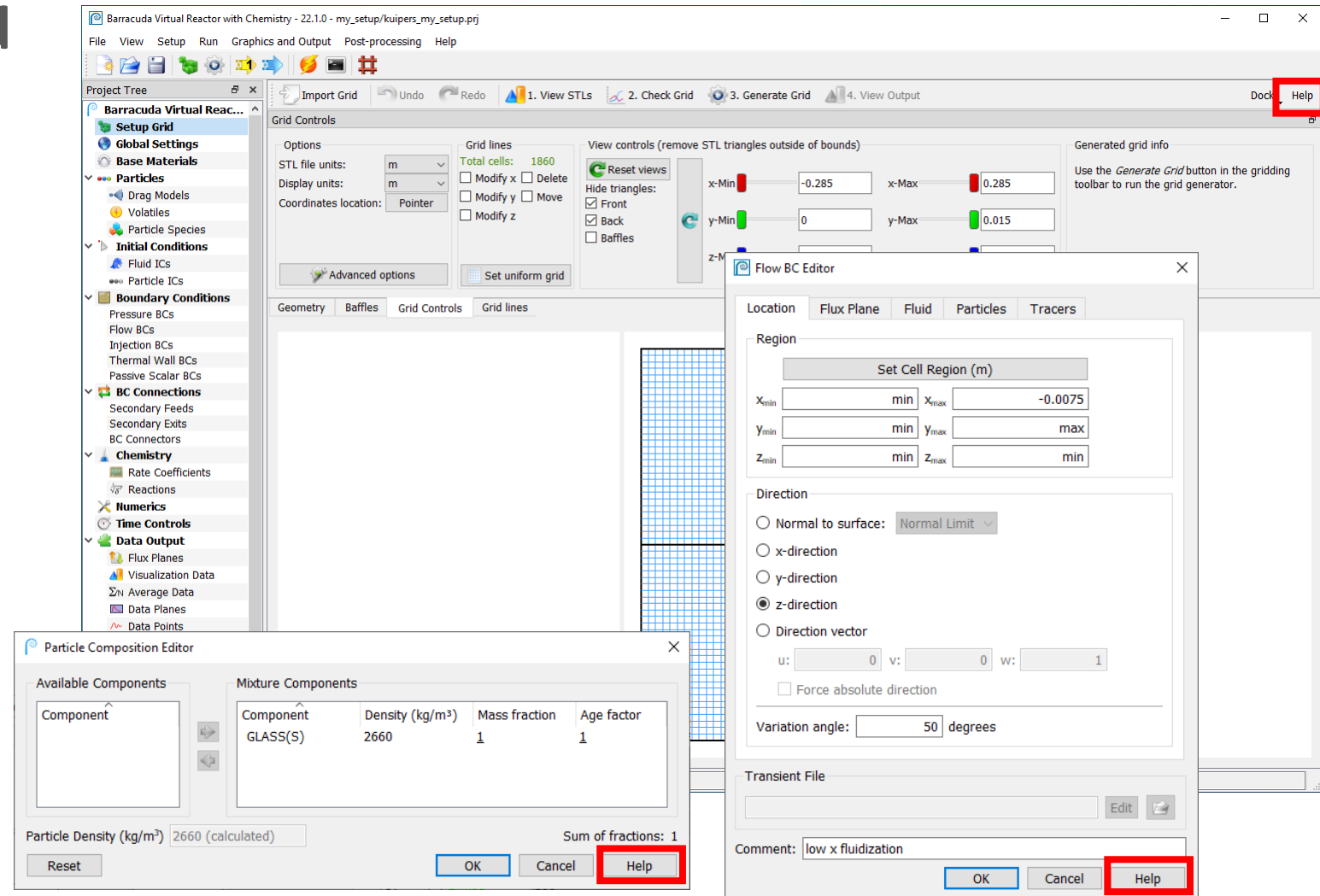
Case 2: Modified geometry with larger cyclone inlet width



# How to Get More Information

All training materials have a limited amount of information about Barracuda and the GUI.

If you want to learn more, click on the Help button in the relevant dialog. This brings up the corresponding section of the User Manual.



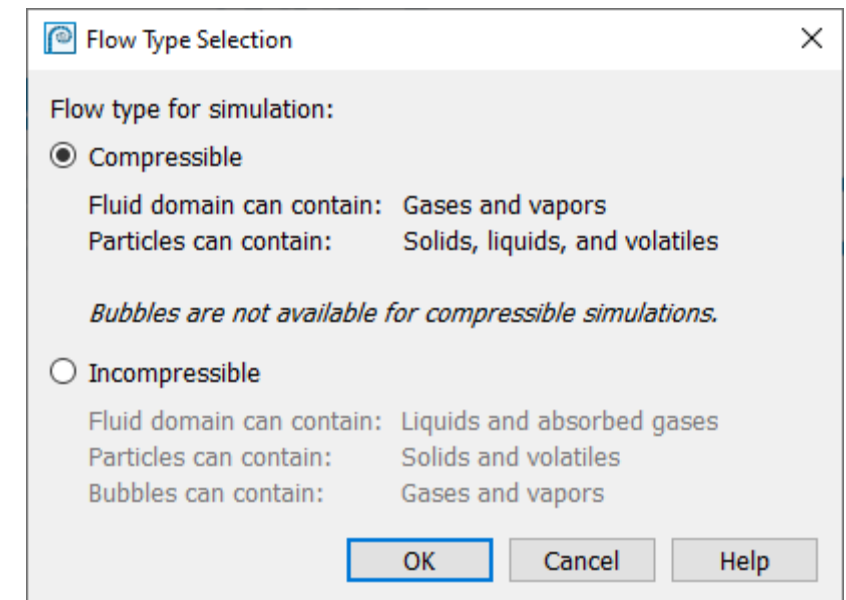
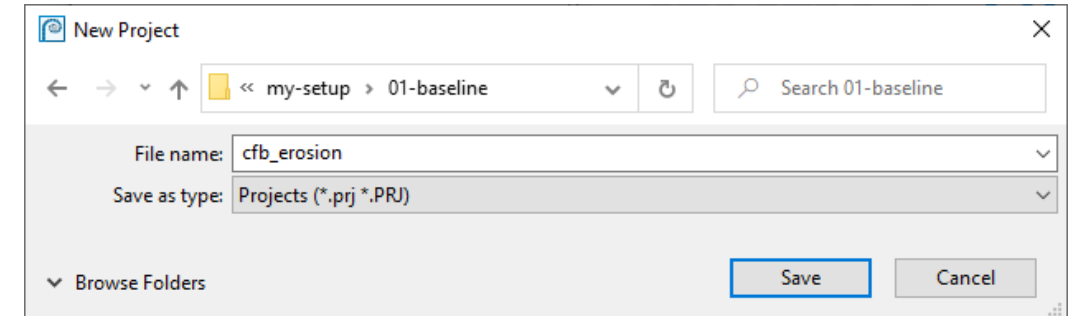
# Project File

Make a new compressible project file in the supplemental training directory:

`\cfb_erosion\my-setup\01-baseline`

With the project name:

`cfb_erosion.prj`



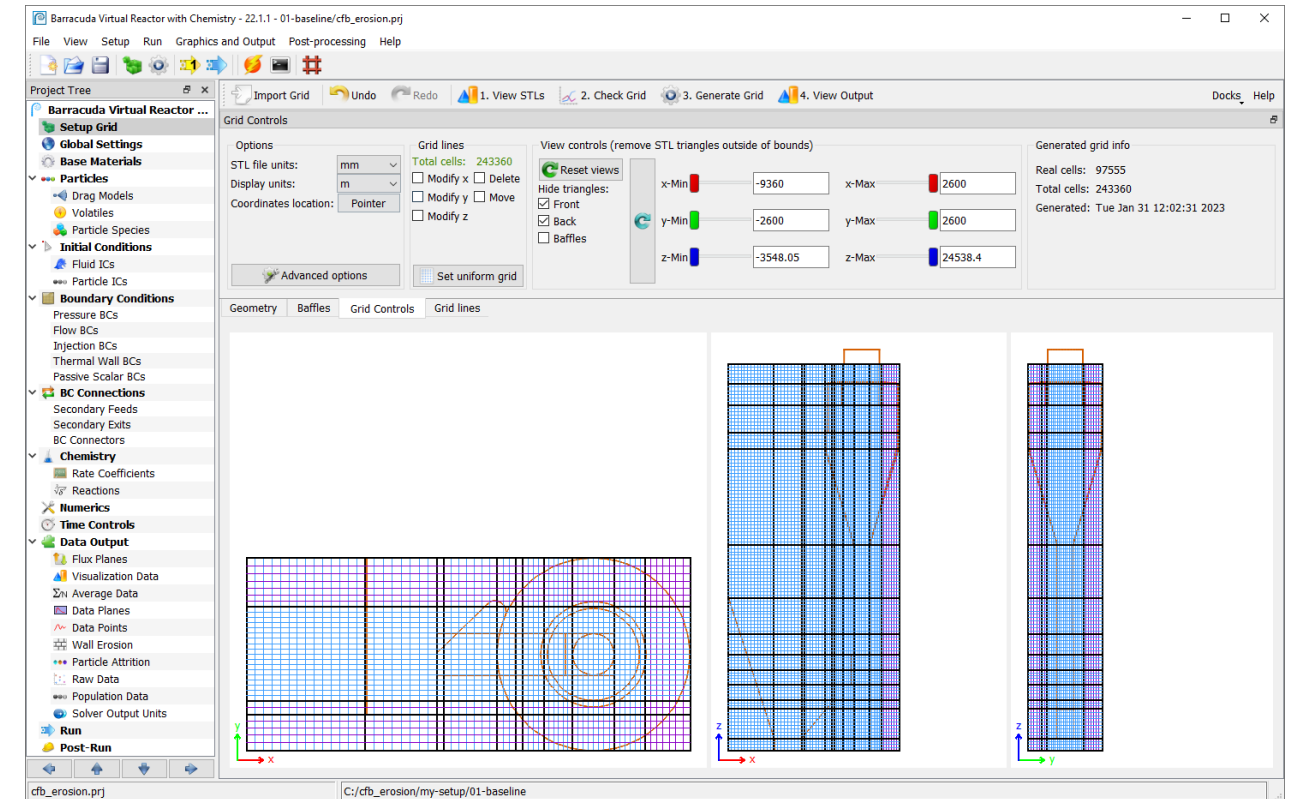
# Setup Grid

## Geometry tab:

- Add original\_baseline\_1150mm.stl

## Grid Controls tab:

- Set STL file units to mm
- Follow along with Non-linear Gridding video: <https://cpfd-software.com/non-linear-gridding-demo/>



# Global Settings

Set Gravity in the z-direction

Select Isothermal at 1123.15 K

Global Settings

Flow Type: Compressible

Fluid domain can contain: Gases and vapors  
Particles can contain: Solids, liquids, and volatiles

*Bubbles are not available for compressible simulations.*

Gravity Settings

x-direction:  m/s<sup>2</sup>    y-direction:  m/s<sup>2</sup>    z-direction:  m/s<sup>2</sup>

Thermal Settings

☒ Isothermal  
Temperature:  K

☐ Thermal

Heat Transfer Coefficients

Radiation Model

☒ None    ☐ Near wall    ☐ P-1    ☒ Cap exposed particle area

Temperature Warning Limits

Minimum:  K    Maximum:  K

☐ Record minimum and maximum temperatures in MinMaxTemp.data log file

Simulation Start Options

Help

# Base Materials

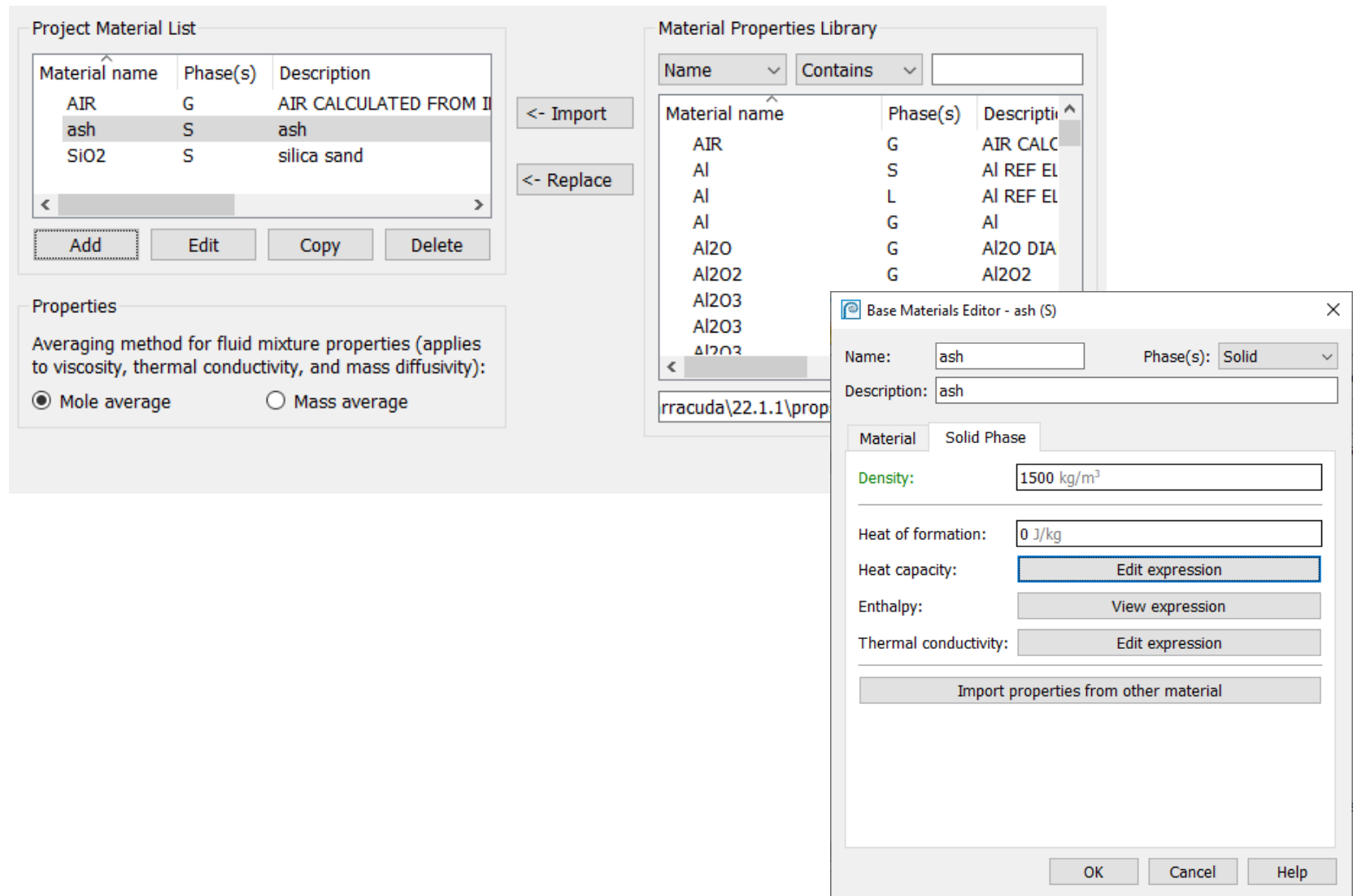
## Import AIR

## Add SiO<sub>2</sub>

- Phase: Solid
- MW = 1 g/mol
- Density = 2650 kg/m<sup>3</sup>

## Add Ash

- Phase: Solid
- MW = 1 g/mol
- Density = 1500 kg/m<sup>3</sup>



The screenshot displays the software interface for managing materials. It includes a 'Project Material List' table, a 'Material Properties Library' table, and a 'Base Materials Editor' dialog for the material 'ash (S)'.

**Project Material List**

Material name	Phase(s)	Description
AIR	G	AIR CALCULATED FROM I
ash	S	ash
SiO2	S	silica sand

Buttons: Add, Edit, Copy, Delete

**Properties**

Averaging method for fluid mixture properties (applies to viscosity, thermal conductivity, and mass diffusivity):

☒ Mole average ☐ Mass average

**Material Properties Library**

Name	Contains
AIR	G
Al	S
Al	L
Al	G
Al2O	G
Al2O2	G
Al2O3	G
Al2O3	G
Al2O3	G

Buttons: <- Import, <- Replace

**Base Materials Editor - ash (S)**

Name: ash Phase(s): Solid

Description: ash

**Material** **Solid Phase**

Density: 1500 kg/m<sup>3</sup>

Heat of formation: 0 J/kg

Heat capacity: Edit expression

Enthalpy: View expression

Thermal conductivity: Edit expression

Import properties from other material

Buttons: OK, Cancel, Help

# Particles

**Set Close pack volume fraction: 0.55**

**Normal-to-wall momentum retention: 0.3**

**Tangent-to-wall momentum retention: 0.995**

**Diffuse bounce: 5**

Particles

Contact and Collision Models

Close pack volume fraction:

Maximum momentum redirection from collision:

☐ Blended acceleration model for the contact force

☒ Transfer liquid mass on collision

Stress Model Options

Wall Interactions

Normal-to-wall momentum retention:

Tangent-to-wall momentum retention:

Diffuse bounce:

Cloud Options

☒ Allow clouds to represent fractional particles

Dense Fluid Forces

☐ Enable virtual mass force

☐ Enable lift force

Help



# Particles Species

**Create the following particle species:**

- Sand, 100% SiO<sub>2</sub>, psd\_sand.sff, WenYu-Ergun drag model
- Fly Ash, 100% ash, psd\_fly\_ash.sff, WenYu-Ergun drag model
- Bottom Ash, 100% ash, psd\_bottom\_ash.sff, WenYu-Ergun drag model

**PSD files are included in the supplemental training directory.**

Particle Species Manager							
Species-ID	Comment	Materials	Size	Sphericity	Emissivity	Drag model	Agglomeration
001	Sand	SiO <sub>2</sub>	psd_sand.sff	1	1	WenYu-Ergun	Off
002	Fly Ash	ash	psd_fly_ash.sff	1	1	WenYu-Ergun	Off
003	Bottom Ash	ash	psd_bottom_ash.sff	1	1	WenYu-Ergun	Off

Add

Edit

Copy

Delete

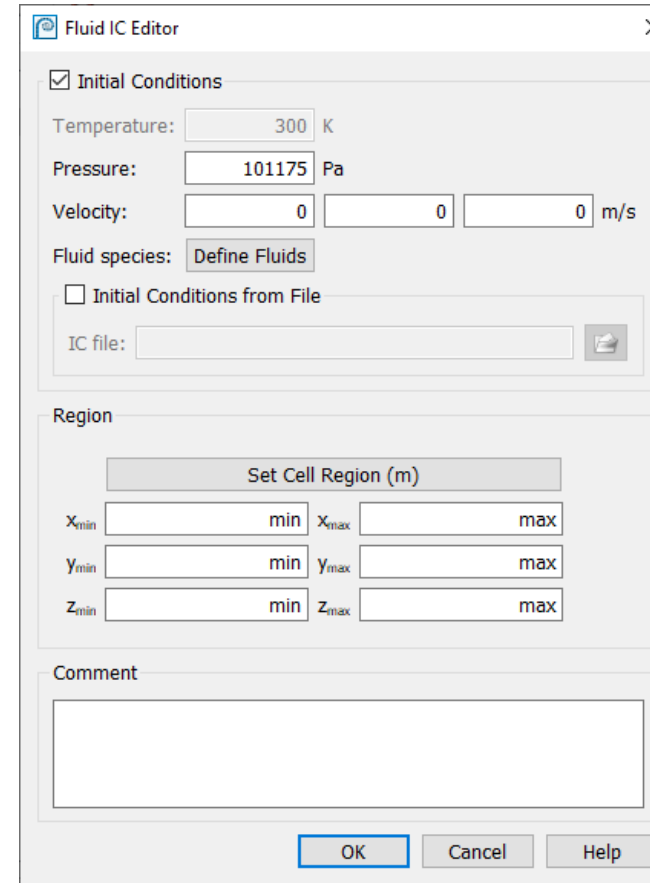
# Fluid and Particle ICs

## Fluid IC

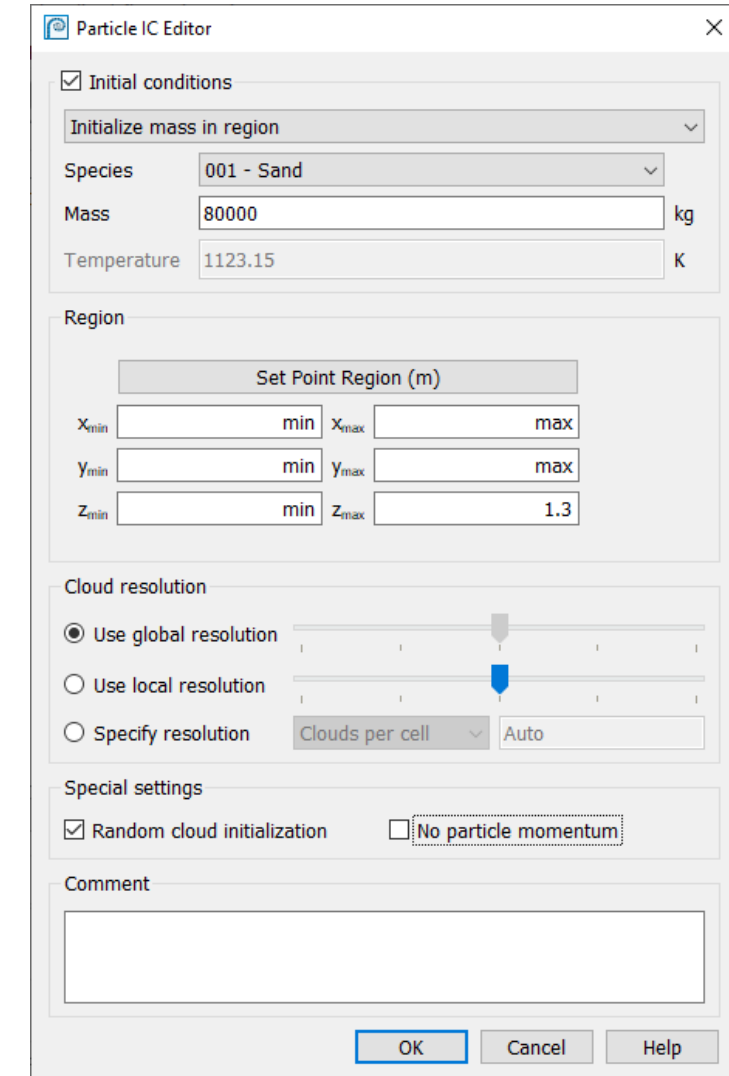
- Pressure = 101175 Pa
- Fluid species = 100% AIR

## Particle IC

- Initialize mass in region
- Species = Sand
- Mass = 80,000 kg
- Set region to min, max for x- and y-, and z-max to 1.3 m



The Fluid IC Editor dialog box is shown. It has a title bar with a close button. The 'Initial Conditions' section is checked, showing Temperature: 300 K, Pressure: 101175 Pa, and Velocity: 0 0 0 m/s. The 'Fluid species' is set to 'Define Fluids'. There is an unchecked checkbox for 'Initial Conditions from File' and an 'IC file' input field. The 'Region' section has a 'Set Cell Region (m)' button and input fields for x\_min, x\_max, y\_min, y\_max, z\_min, and z\_max, all set to 'min' or 'max'. A 'Comment' text area is at the bottom, along with 'OK', 'Cancel', and 'Help' buttons.



The Particle IC Editor dialog box is shown. It has a title bar with a close button. The 'Initial conditions' section is checked, with a dropdown set to 'Initialize mass in region'. The 'Species' is '001 - Sand', 'Mass' is 80000 kg, and 'Temperature' is 1123.15 K. The 'Region' section has a 'Set Point Region (m)' button and input fields for x\_min, x\_max, y\_min, y\_max, z\_min, and z\_max, with z\_max set to 1.3. The 'Cloud resolution' section has radio buttons for 'Use global resolution' (selected), 'Use local resolution', and 'Specify resolution', with a slider and 'Clouds per cell' dropdown. The 'Special settings' section has checked boxes for 'Random cloud initialization' and 'No particle momentum'. A 'Comment' text area is at the bottom, along with 'OK', 'Cancel', and 'Help' buttons.

# Pressure BC

Add a pressure BC with information shown here for cyclone outlet.

Use 100% AIR for Applied fluids

Pressure Boundary Conditions Editor

	Time (s)	Pressure (Pa)	Temperature (K)	Area Fraction	Particle Feed	K-Factor
1	0	101175	1123.15	1	<input type="radio"/> Off	0
2					<input checked="" type="radio"/> On	

File:

Pressure BC Editor

Location Flux Plane Fluid Particles Tracers

Region

Select region (m)

X<sub>min</sub>  X<sub>max</sub>

Y<sub>min</sub>  Y<sub>max</sub>

Z<sub>min</sub>  Z<sub>max</sub>

Direction

☐ x-direction  
☐ y-direction  
☒ z-direction

Flow Conditions

☒ Transient file:

☐ Specify values:

Area fraction:

Pressure:  Pa

Temperature:  K

K-factor:

Fluid Composition

Fluid inflow properties:

Applied fluids:

Comment:

Pressure BC Editor

Location Flux Plane Fluid Particles Tracers

Name:

Fluid species behavior:

☒ Bin by particle size  bins

☐ Output raw particle data

☐ Output tracer data

Behavior at Boundary

☐ No outflow

☒ Outflow with size filtering:

Minimum:

Maximum:

Units:

☐ Feed specified as volume fraction

☐ Feed specified as mass flux

☐ Feed specified as mass flow rate

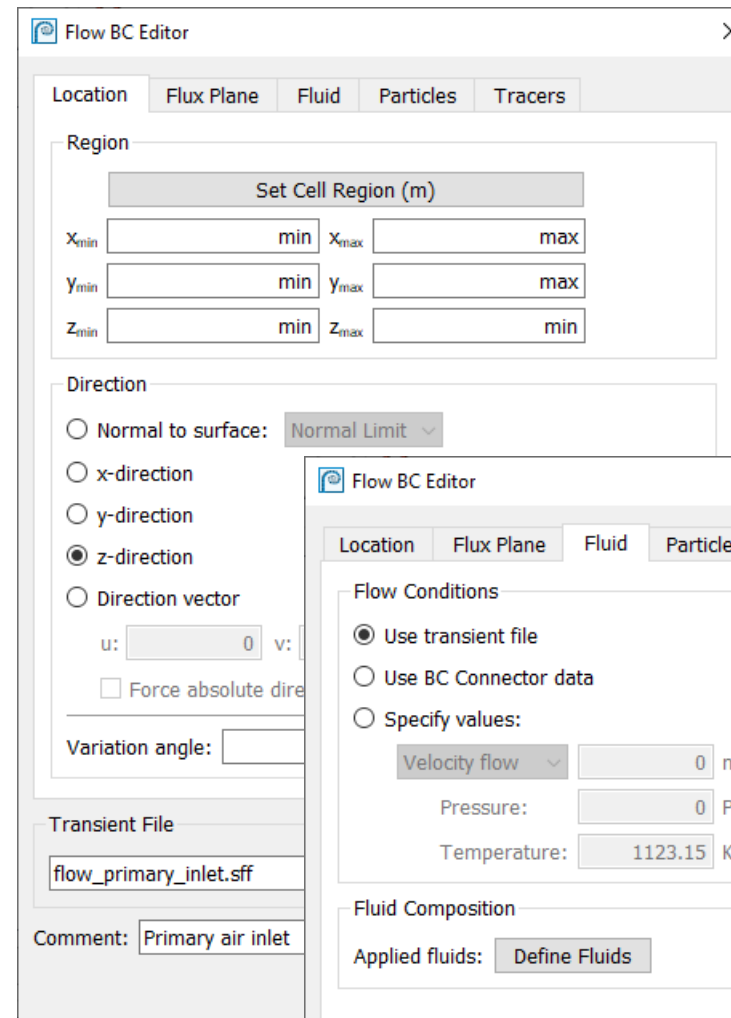
Feed Settings

Comment:

# Flow BCs

Add flow BC with information shown here for primary air inlet.

Use 100% AIR for Applied fluids



Flow BC Editor

Location Flux Plane Fluid Particles Tracers

Region

Set Cell Region (m)

$x_{min}$  min  $x_{max}$  max

$y_{min}$  min  $y_{max}$  max

$z_{min}$  min  $z_{max}$  min

Direction

☐ Normal to surface: Normal Limit

☐ x-direction

☐ y-direction

☒ z-direction

☐ Direction vector

u: 0 v: 0

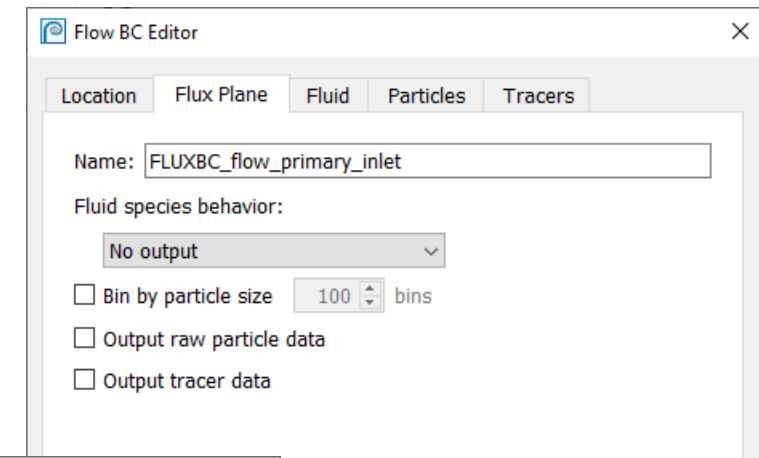
☐ Force absolute direction

Variation angle: 0

Transient File

flow\_primary\_inlet.sff

Comment: Primary air inlet



Flow BC Editor

Location Flux Plane Fluid Particles Tracers

Name: FLUXBC\_flow\_primary\_inlet

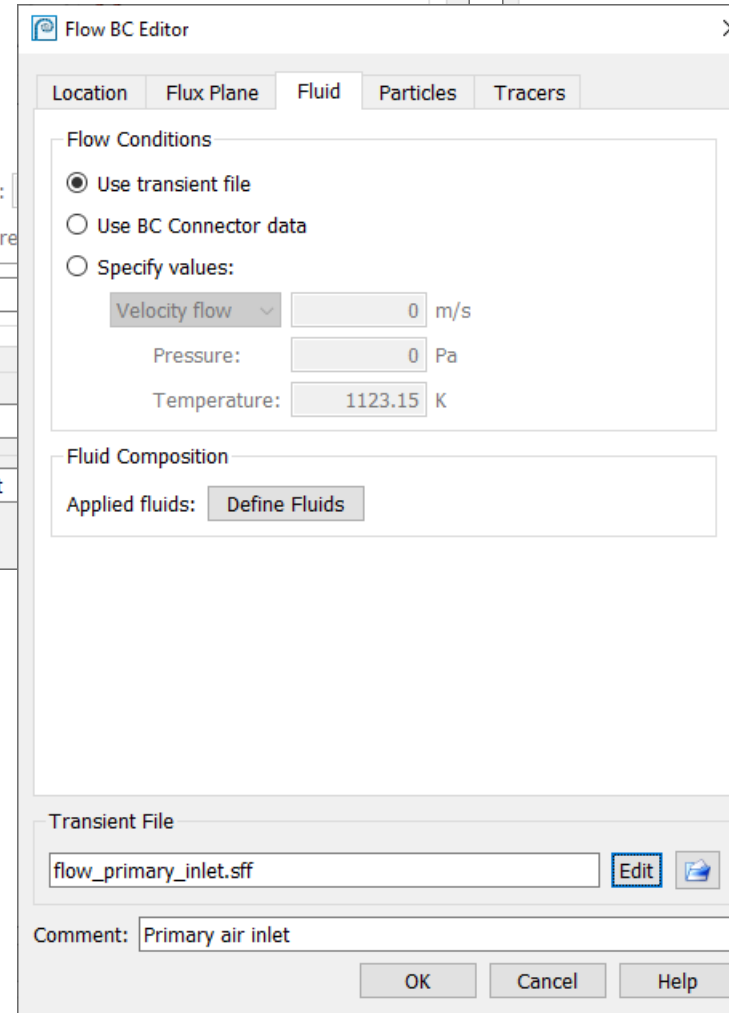
Fluid species behavior:

No output

☐ Bin by particle size 100 bins

☐ Output raw particle data

☐ Output tracer data



Flow BC Editor

Location Flux Plane Fluid Particles Tracers

Flow Conditions

☒ Use transient file

☐ Use BC Connector data

☐ Specify values:

Velocity flow 0 m/s

Pressure: 0 Pa

Temperature: 1123.15 K

Fluid Composition

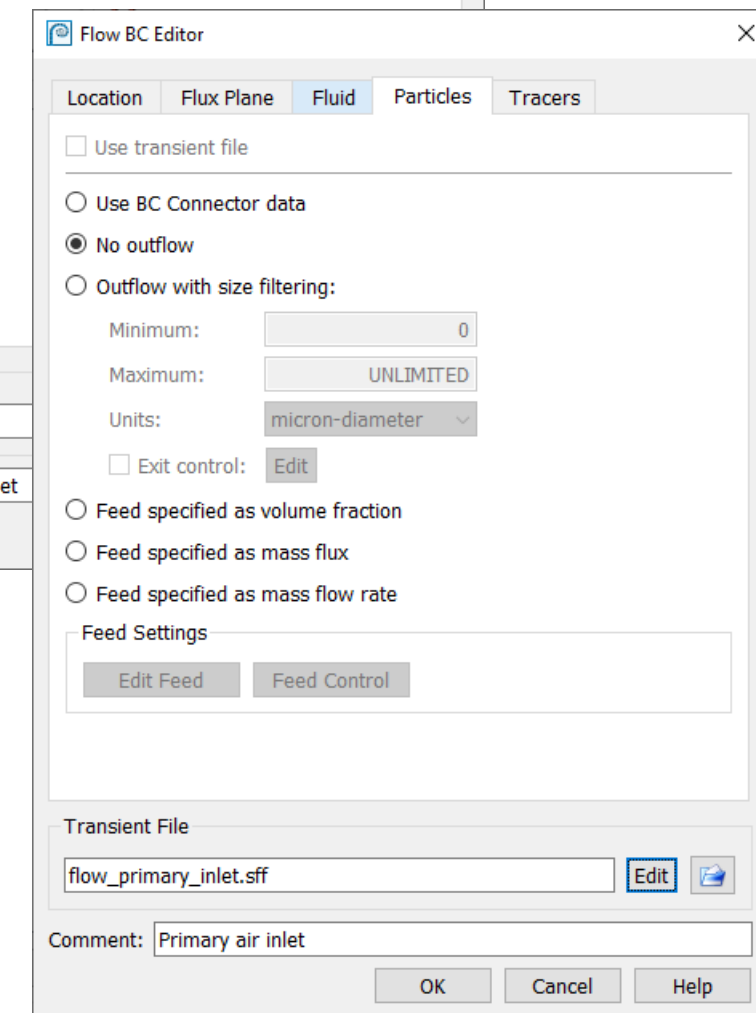
Applied fluids: Define Fluids

Transient File

flow\_primary\_inlet.sff

Comment: Primary air inlet

OK Cancel Help



Flow BC Editor

Location Flux Plane Fluid Particles Tracers

☐ Use transient file

☐ Use BC Connector data

☒ No outflow

☐ Outflow with size filtering:

Minimum: 0

Maximum: UNLIMITED

Units: micron-diameter

☐ Exit control: Edit

☐ Feed specified as volume fraction

☐ Feed specified as mass flux

☐ Feed specified as mass flow rate

Feed Settings

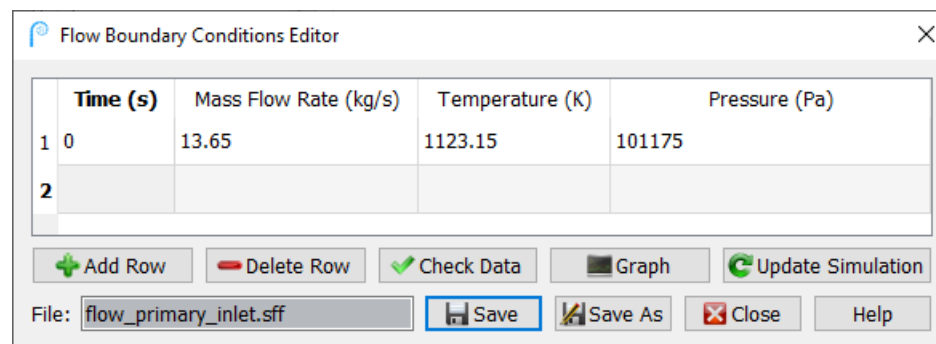
Edit Feed Feed Control

Transient File

flow\_primary\_inlet.sff

Comment: Primary air inlet

OK Cancel Help



Flow Boundary Conditions Editor

	Time (s)	Mass Flow Rate (kg/s)	Temperature (K)	Pressure (Pa)
1	0	13.65	1123.15	101175
2				

+ Add Row - Delete Row ✓ Check Data 📊 Graph ↻ Update Simulation

File: flow\_primary\_inlet.sff Save Save As Close Help

# Flow BCs

Add flow BCs for 4 secondary air inlets.

Use information in the table to set the cell region and Flux Plane Name for each flow BC.

The Flow BC Editor dialog box has tabs for Location, Flux Plane, Fluid, Particles, and Tracers. The Region section includes a 'Set Cell Region (m)' button and input fields for x<sub>min</sub>, x<sub>max</sub>, y<sub>min</sub>, y<sub>max</sub>, z<sub>min</sub>, and z<sub>max</sub>. The Direction section has radio buttons for 'Normal to surface', 'x-direction', 'y-direction', 'z-direction', and 'Direction vector', along with a 'Normal Limit' dropdown and input fields for u, v, and w. A 'Force absolute direction' checkbox and a 'Variation angle' field are also present. The Transient File section has a text box for the file name, an 'Edit' button, and a 'Comment' field.

The Flow BC Editor dialog box shows the 'Fluid species behavior' section with a dropdown menu set to 'No output'. Below this are checkboxes for 'Bin by particle size' (with a '100 bins' input), 'Output raw particle data', and 'Output tracer data'. The Transient File section is identical to the previous dialog, showing the file name 'flow\_secondary\_inlet.sff' and the comment 'secondary inlet 1'.

Secondary Inlet	xmin	xmax	ymin	ymax	zmin	zmax	Flux Plane Name
1	-5.95	-5.52	-1.63	-1.44	1.9	2.32	FLUXBC_flow_secondary_inlet_1
2	-5.95	-5.52	max	max	1.9	2.32	FLUXBC_flow_secondary_inlet_2
3	-6.57	-6.37	1.3	1.58	-2	-1.57	FLUXBC_flow_secondary_inlet_3
4	-6.57	-6.36	-0.53	-0.3	-2	-1.57	FLUXBC_flow_secondary_inlet_4

# Flow BCs

Fill in the information in the numbered order shown here for the Fluid and Particles tabs.

Use 100% AIR for Applied fluids.

Use the same particle species mixture and `flow_secondary_inlet.sff` for all 4 BCs.

The image displays three screenshots of the software interface for configuring flow boundary conditions, with numbered annotations (1-6) indicating the sequence of steps.

**Flow BC Editor (Left):**

- 1** (Flow Conditions): ☒ Use transient file
- 2** (Fluid Composition): Applied fluids: **Define Fluids**
- Transient File:** `flow_secondary_inlet.sff`, Comment: `secondary inlet 1`

**Flow BC Editor (Right):**

- 4** (Use transient file): ☒ Use transient file
- 3** (Feed specified as mass flow): ☒ Feed specified as mass flow
- 5** (Particle Feed Settings):
  - Particle Species: 001: Sand, 002: Fly Ash (0.66667), 003: Bottom Ash (0.33333)
  - Particle feed: Solid fraction (Mass Fraction), Particle/fuid slip ratio (0.5), Particle feed per average volume (125), Particle feed mass flow rate (0)
- 6** (Transient File): `flow_secondary_inlet.sff`, Comment: `secondary inlet 1`

**Flow Boundary Conditions Editor (Bottom):**

	Time (s)	Mass Flow Rate (kg/s)	Temperature (K)	Pressure (Pa)	Particle Feed	Number Density Manual	Particle Slip	Particle Mass Flow Rate (kg/s)
1	0	4.8514	1123.15	101175	<input checked="" type="radio"/> On	2000	1	0.1736
2					<input checked="" type="radio"/> On			

Buttons: + Add Row, - Delete Row, ✓ Check Data, Graph, Update Simulation

File: `flow_secondary_inlet.sff` | Save | Save As | Close | Help

# Flow BCs

Add flow BC with information shown here for seal pot air inlet.

Use Set Cell Region to graphically select the bottom surface of the seal pot.

Use 100% AIR for Applied fluids

The image displays four screenshots of the software's configuration windows for a flow boundary condition (BC) named 'FLUXBC\_flow\_seal\_pot'.

- Top Left Screenshot (Flow BC Editor - Location Tab):** Shows the 'Set Cell Region (m)' with coordinates:  $x_{min} = -1.96651$ ,  $x_{max} = 0.820299$ ,  $y_{min} = -0.726217$ ,  $y_{max} = 0.74345$ ,  $z_{min} = -0.164238$ , and  $z_{max} = 0.255995$ . The 'Direction' is set to 'z-direction'.
- Top Right Screenshot (Flow BC Editor - Fluid Tab):** Shows 'Fluid species behavior' set to 'No output'. There are checkboxes for 'Bin by particle size' (100 bins), 'Output raw particle data', and 'Output tracer data', all of which are currently unchecked.
- Middle Screenshot (Flow BC Editor - Fluid Tab):** Shows 'Flow Conditions' with 'Use transient file' selected. The 'Specify values' section shows 'Velocity flow' set to 0 m/s, 'Pressure' set to 0 Pa, and 'Temperature' set to 1123.15 K. The 'Fluid Composition' section shows 'Applied fluids' set to 'Define Fluids'.
- Bottom Left Screenshot (Flow Boundary Conditions Editor):** A table showing the transient data for the flow BC. The table has four columns: Time (s), Mass Flow Rate (kg/s), Temperature (K), and Pressure (Pa). The first row shows data at Time 0.

	Time (s)	Mass Flow Rate (kg/s)	Temperature (K)	Pressure (Pa)
1	0	0.64625	1123.15	101175
2				

The bottom right screenshot shows the 'Feed Settings' section with 'Edit Feed' and 'Feed Control' buttons. The 'Transient File' is set to 'flow\_seal\_pot.sff' and the 'Comment' is 'Seal pot inlet'.

# Time Controls

## Set Time step and End time

### Time Controls

This section allows configuration of the time step size to take during a period of time for the calculation. Only the first row is required. Subsequent rows can be entered to have different time steps for different time periods. For example, starting the calculation at a smaller time step is recommended, and then increasing the time step for rows 2-5 over simulation time.

#### Time step and duration settings

	Time step		End time	
1.	<input type="text" value="0.002"/>	s	<input type="text" value="40"/>	s
2.	<input type="text"/>	s	<input type="text"/>	s
3.	<input type="text"/>	s	<input type="text"/>	s
4.	<input type="text"/>	s	<input type="text"/>	s
5.	<input type="text"/>	s	<input type="text"/>	s

[Advanced time step settings](#)

#### Restart file intervals

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

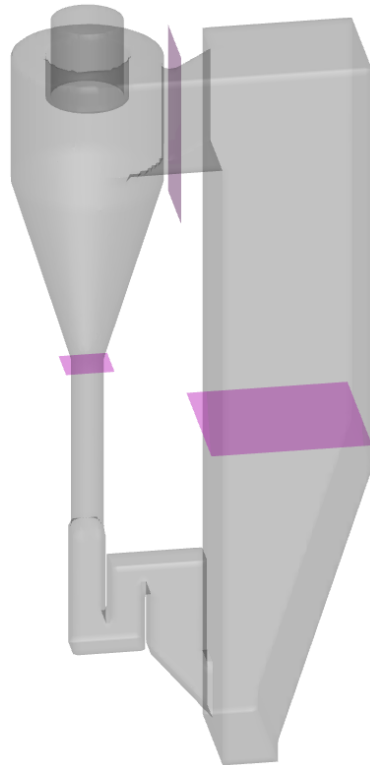
Backtrack interval (IC\_)  realtime minutes

[Help](#)



# Flux Planes

Define 3 internal flux planes to track flow:  
cyclone inlet, dipleg, and  
above bed



Flux Plane Editor

Location

Normal Direction: x

Select plane (m)

x: -2.9

y<sub>min</sub>: 0 y<sub>max</sub>: max

z<sub>min</sub>: 17 z<sub>max</sub>: 23

Flux plane options

Name: FLUX\_cyclone\_inlet

Fluid species behavior: No output

☐ Bin by particle size 100 bins

☐ Output raw particle data

☐ Output tracer data

☐ Reset particle residence time

☐ Reset tracer residence time

☐ Directional flux

Comment

Cyclone inlet

OK Cancel Help

Flux Plane Editor

Location

Normal Direction: z

Select plane (m)

z: 10.45

x<sub>min</sub>: -0.8 x<sub>max</sub>: 0.78

y<sub>min</sub>: -0.7 y<sub>max</sub>: 0.7

Flux plane options

Name: FLUX\_cyclone\_dipleg

Fluid species behavior: No output

☐ Bin by particle size 100 bins

☐ Output raw particle data

☐ Output tracer data

☐ Reset particle residence time

☐ Reset tracer residence time

☐ Directional flux

Comment

Cyclone dipleg

OK Cancel Help

Flux Plane Editor

Location

Normal Direction: z

Select plane (m)

z: 8

x<sub>min</sub>: min x<sub>max</sub>: -4

y<sub>min</sub>: min y<sub>max</sub>: max

Flux plane options

Name: FLUX\_above\_bed

Fluid species behavior: No output

☐ Bin by particle size 100 bins

☐ Output raw particle data

☐ Output tracer data

☐ Reset particle residence time

☐ Reset tracer residence time

☐ Directional flux

Comment

Above bed

OK Cancel Help

# Visualization Data Output

Set Output file interval and select the options shown for Visualization Data Output.

Visualization Data Output

Output formats  
☒ Tecplot (\*.plt files) ☐ GMV (Gmv.\* files)

Output file interval  
Plot interval:  s Number of files produced using current end time of **40s**: **161**

Cell Data

Available Data		Selected Data
Cell volume		Cell indices
Clouds per cell		<b>CFL</b>
Convective wall heat transfer		dp/dz
dp/dx	➡	Fluid density
dp/dy	➡	Fluid mass flux
Dynamic pressure		Fluid velocity
Fluid composition mass concentration		Particle mass flux
Fluid composition mass fraction		Particle velocity
Fluid composition mole concentration		Particle volume fraction
Fluid composition mole fraction		Pressure

Particle Data

Available Data		Selected Data
Cell ID		Cloud ID
Drag		Cloud mass
Liquid fraction total		Particle density
Liquid mass total		Particle size
P1 radiation flux		Particle species
Particle mass		Particle speed
Particle material		Particle velocity
Particle temperature		Particle volume fraction
<b>Particles per cloud</b>	➡	<b>Residence time</b>

Help

# Wall Erosion

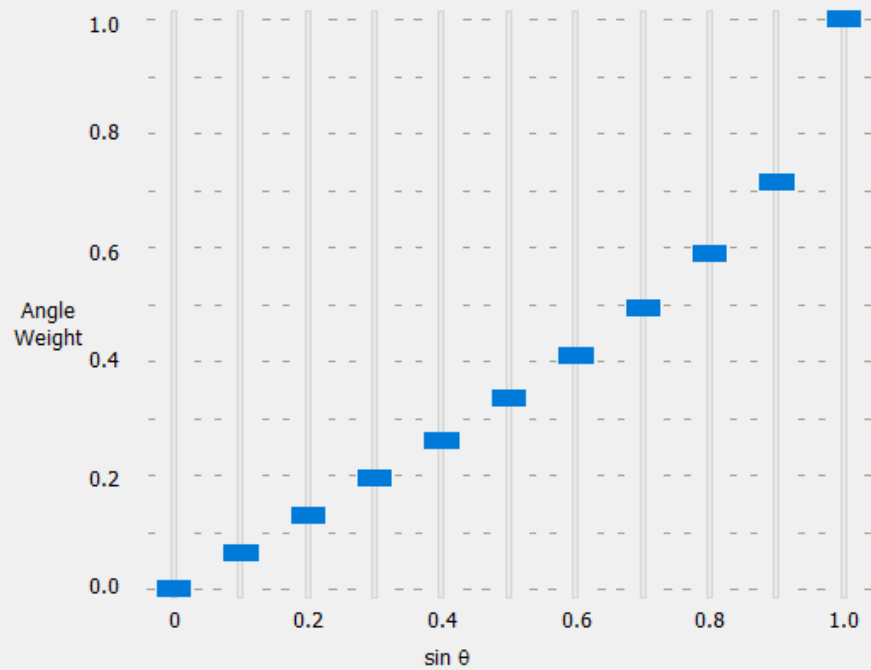
Select Enable Wall Erosion

Set the angular dependence for refractory based on this reference:

<https://cpfd-software.com/how-do-i-set-wall-erosion-parameters-for-my-unit/>

☒ Enable Wall Erosion

Graph input



The graph shows a series of blue rectangular bars representing the angular weight for different values of sin θ. The x-axis is labeled 'sin θ' and ranges from 0 to 1.0. The y-axis is labeled 'Angle Weight' and ranges from 0.0 to 1.0. The bars are positioned at sin θ values of 0, 0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9, and 1.0, with heights corresponding to the values in the table on the right.

θ	sin θ	Angle Weight
0.0°	0.0	0
5.7°	0.1	0.063
11.5°	0.2	0.127
17.5°	0.3	0.194
23.6°	0.4	0.262
30.0°	0.5	0.333
36.9°	0.6	0.410
44.4°	0.7	0.493
53.1°	0.8	0.590
64.2°	0.9	0.713
90.0°	1.0	1

Text input

θ	sin θ	Angle Weight
0.0°	0.0	<input type="text" value="0"/>
5.7°	0.1	<input type="text" value="0.063"/>
11.5°	0.2	<input type="text" value="0.127"/>
17.5°	0.3	<input type="text" value="0.194"/>
23.6°	0.4	<input type="text" value="0.262"/>
30.0°	0.5	<input type="text" value="0.333"/>
36.9°	0.6	<input type="text" value="0.410"/>
44.4°	0.7	<input type="text" value="0.493"/>
53.1°	0.8	<input type="text" value="0.590"/>
64.2°	0.9	<input type="text" value="0.713"/>
90.0°	1.0	<input type="text" value="1"/>

Wear model parameters

Start calculating wear at time:  s

Wear Exponents

$$m^{1.5} \times u^{3.5}$$

Mass exponent  Velocity exponent

Minimum Limit

Limit value

Notes

θ is the angle between the particle vector and wall tangent, i.e.:  
normal: θ=90°  
tangent: θ=0°

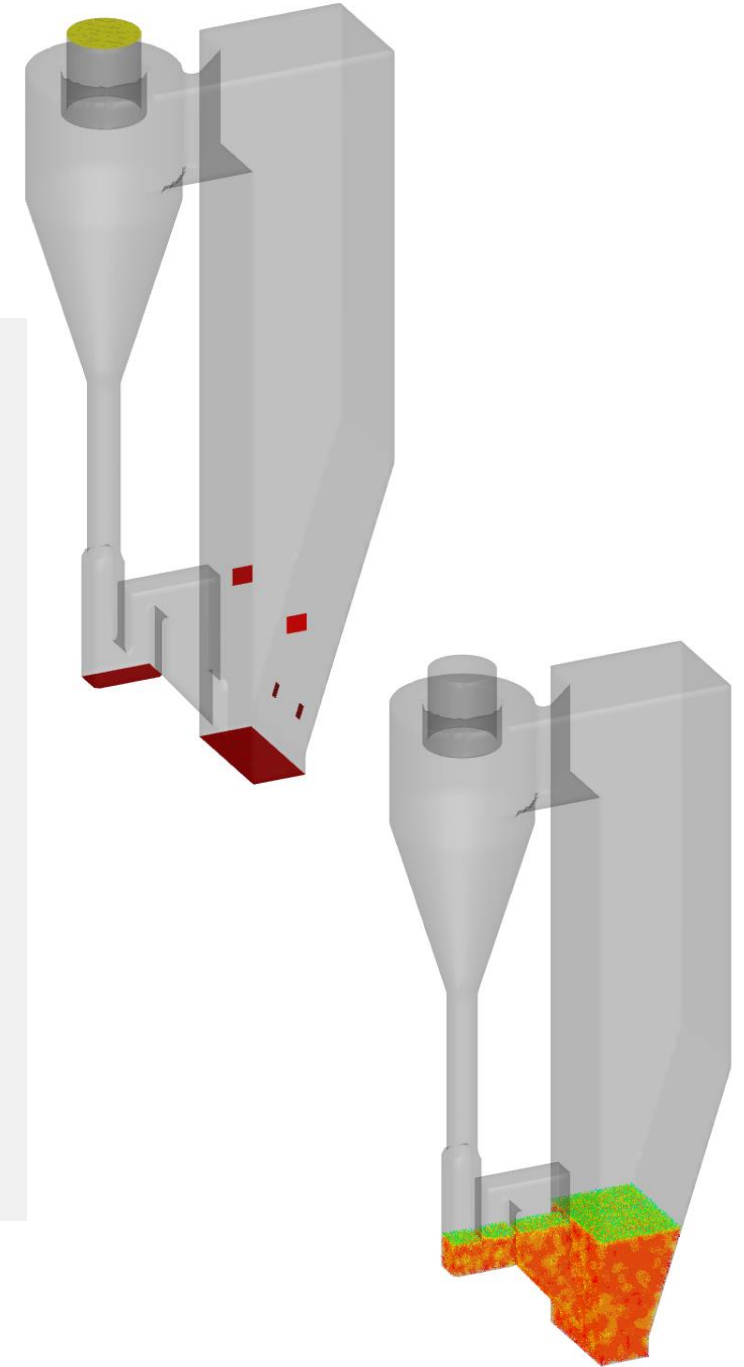
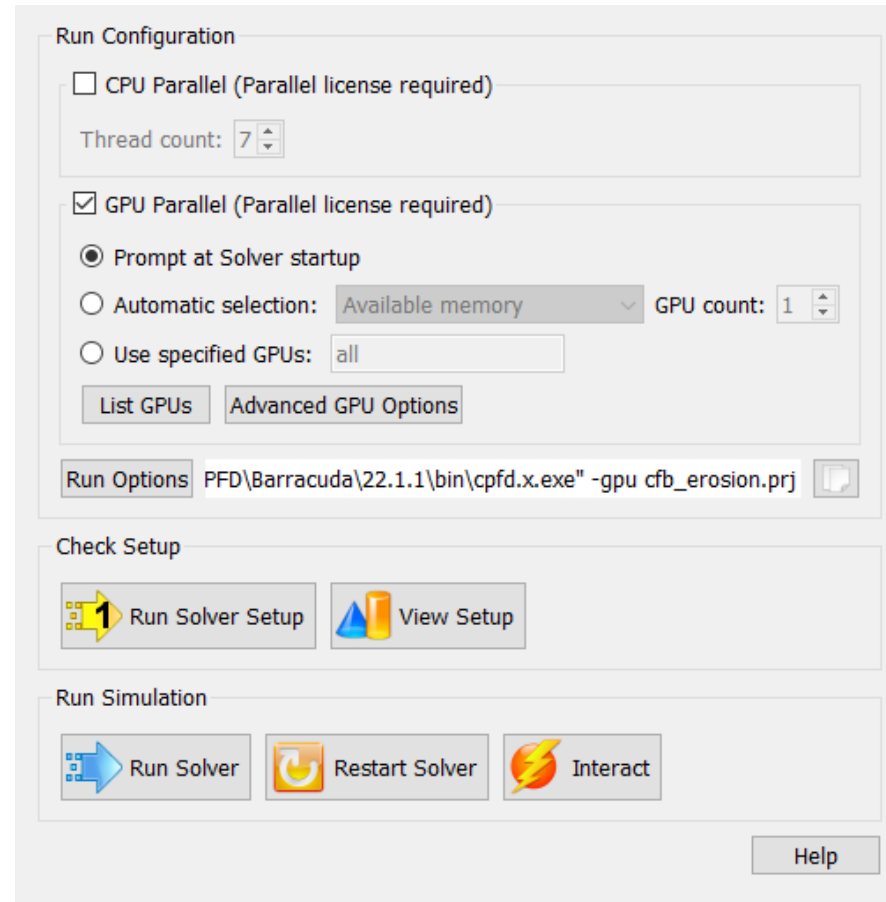
Angle weight is a coefficient of the impact (wear) as a function of θ.

# Run

If NVIDIA GPU card is available,  
select GPU Parallel

Run Solver Setup and View  
Setup

Once you have checked the  
setup, Run Solver to start the  
simulation for Case 1

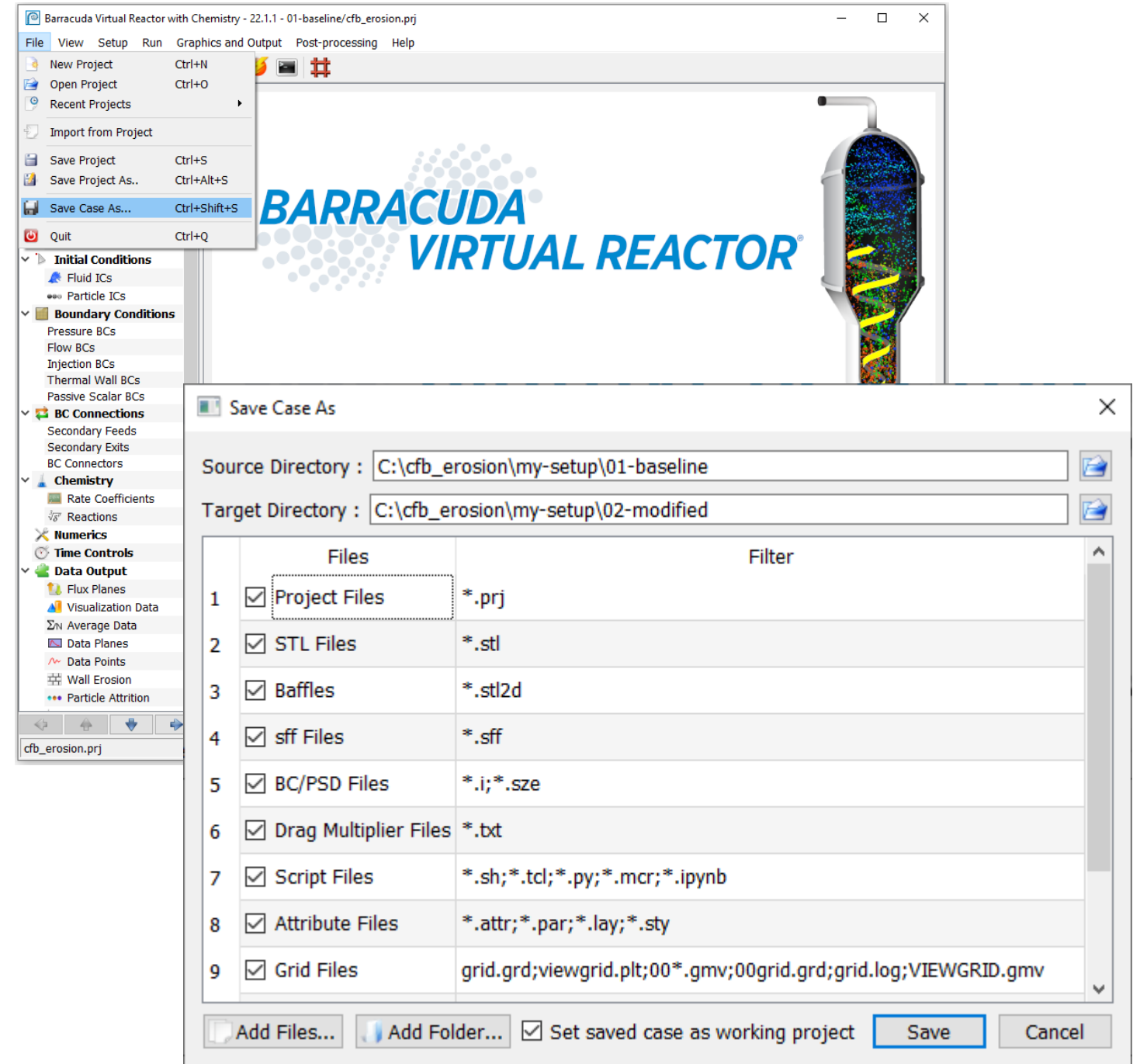


# Save Case As...

The modified geometry project directory can be created from the existing project directory using Save Case As...

Select 02-modified as the target directory

Ensure Files 1-9 are selected and click Save



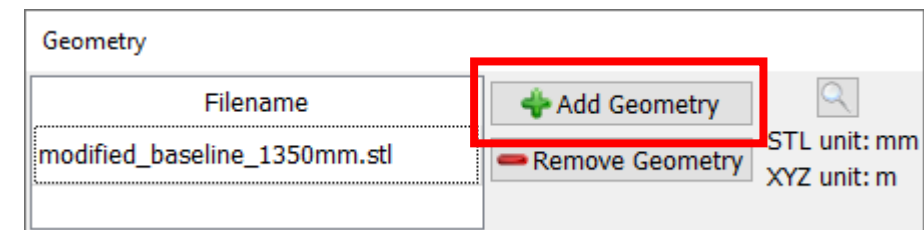
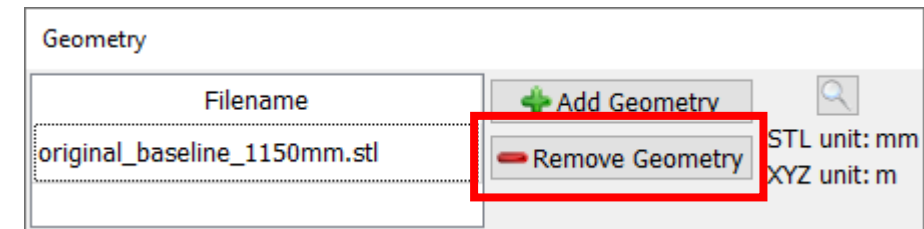
## Case 2: Modified

In the 02-modified project, go to  
Geometry tab of Setup Grid

Highlight original\_baseline\_1150mm.stl  
and Remove Geometry

Add Geometry and select  
modified\_baseline\_1350mm.stl

Generate the grid



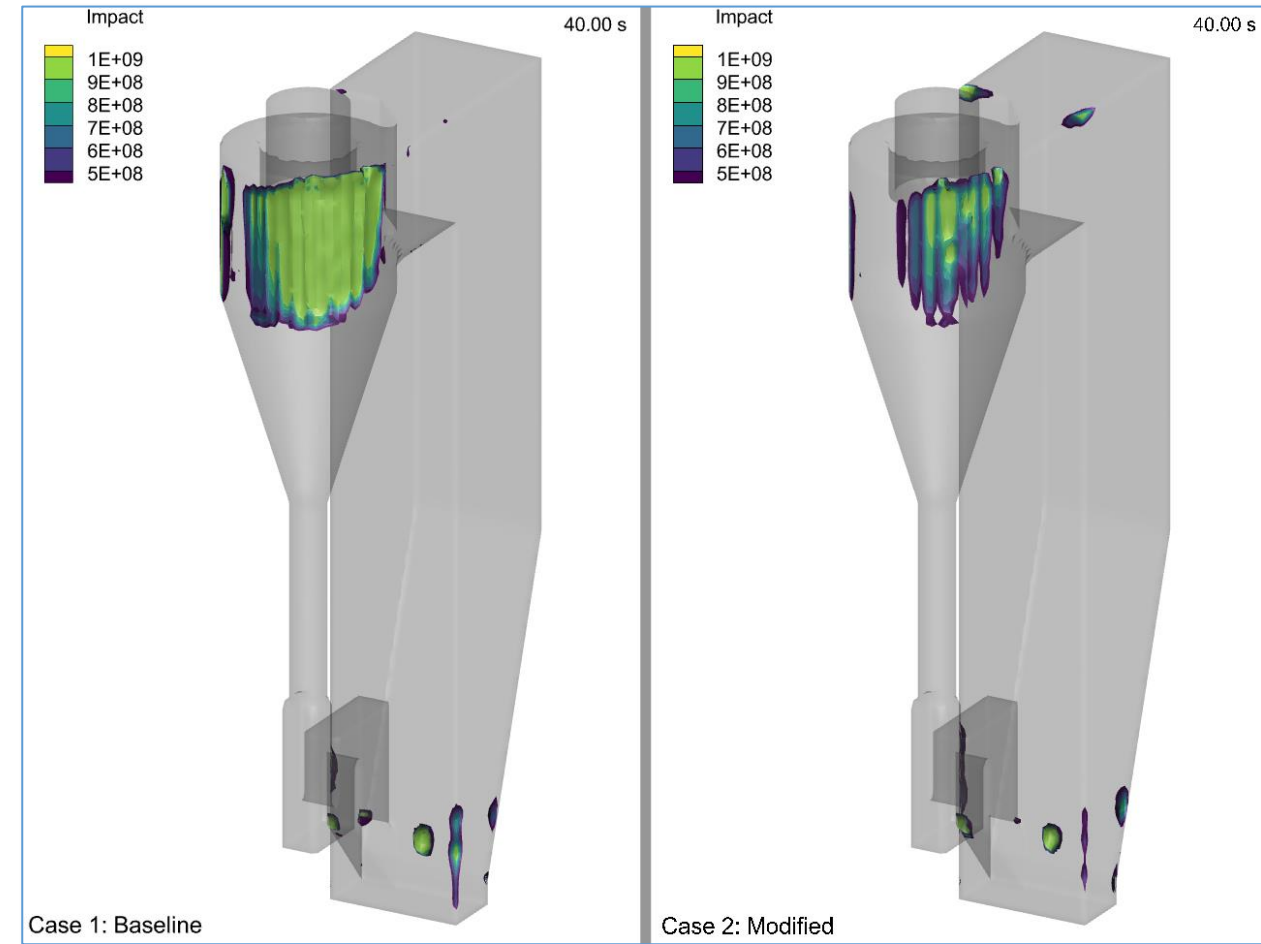




# Post-Processing: Cyclone Erosion

Follow along with this video to create your own view of erosion. Compare the erosion in the cyclone in Case 1 with Case 2.

<https://cpfd-software.com/tecplot-for-barracuda-using-isosurfaces-to-analyze-erosion-results/>





# Post-Processing: Slices

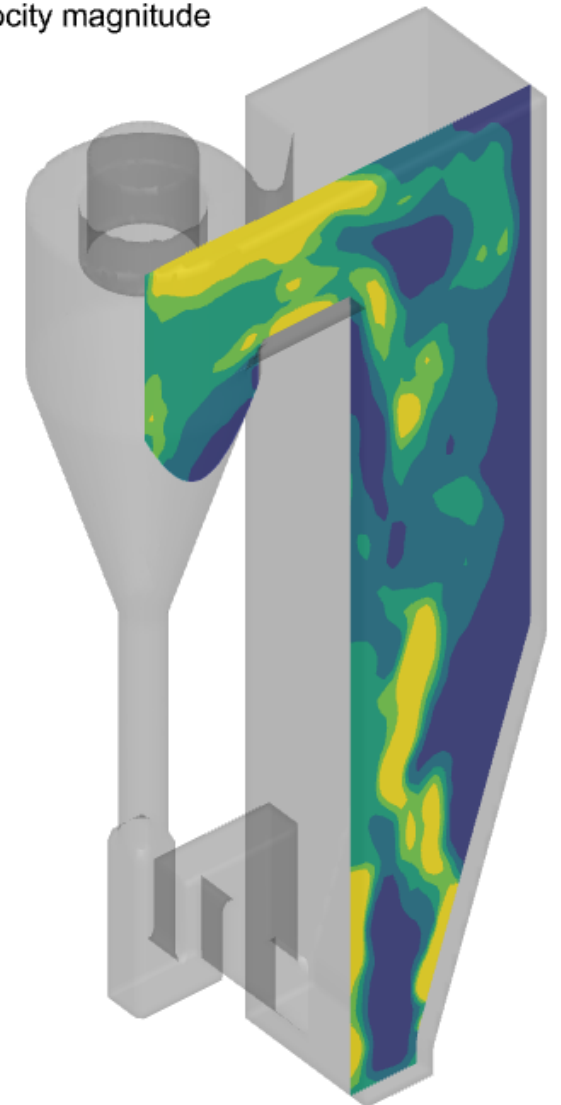
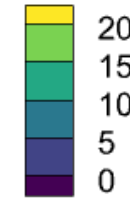
Create a slice colored by fluid velocity magnitude through the center of the cyclone inlet

- Erosion is highly dependent on velocity so it is useful to visualize the velocity in the cyclone inlet tube in both cases

This video shows how to create slices:

<https://cpfd-software.com/tecplot-for-barracuda-calculating-spatial-averages-on-slices/>

Fluid velocity magnitude

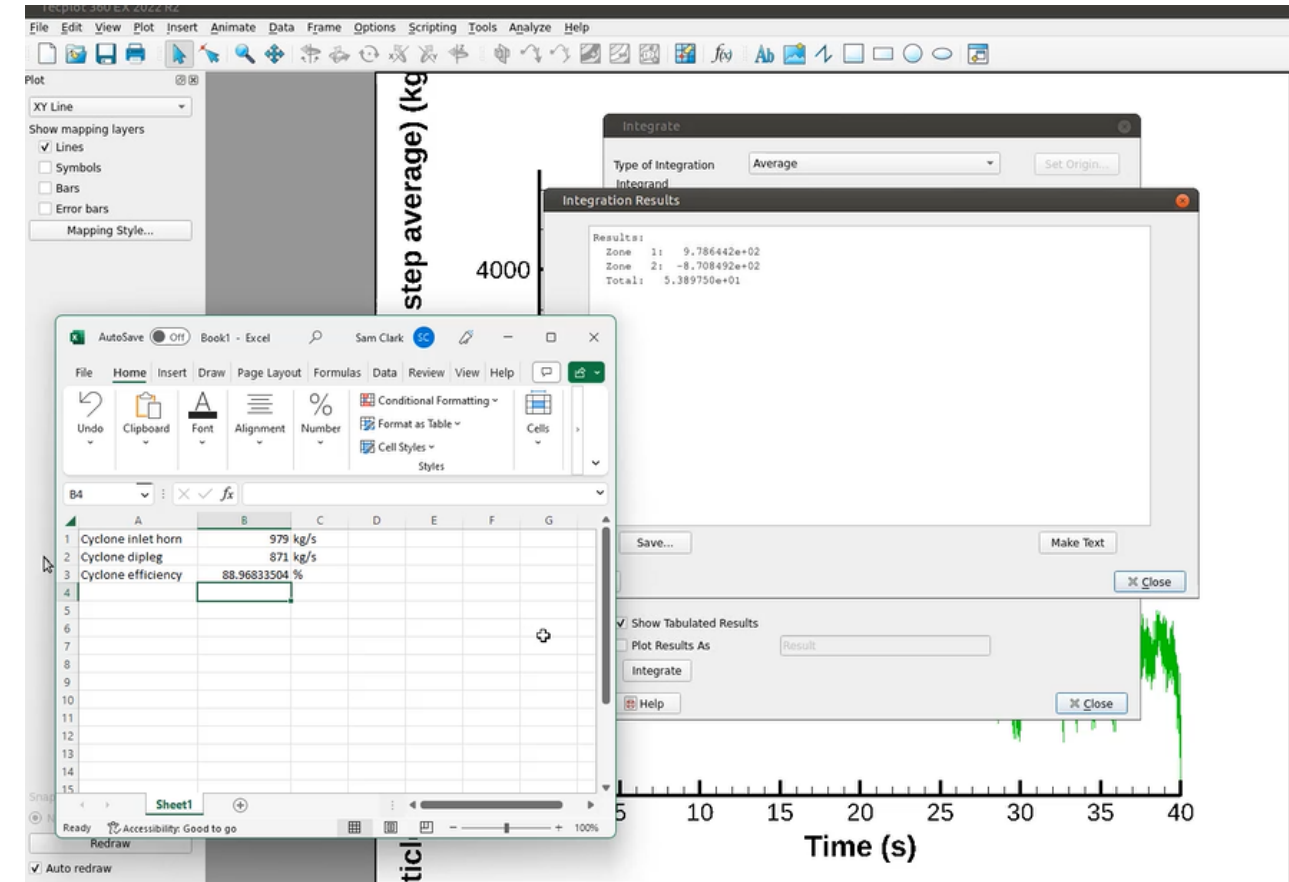


# Post-Processing: Cyclone Efficiency

Follow along with this video to create an xy plot in Tecplot using flux plane data to calculate cyclone efficiency.

- Calculate the efficiency for both cases and compare

<https://cpfd-software.com/tecplot-for-barracuda-calculating-cyclone-efficiency/>



# Summary

This model showed an example of a CFB loop simulation based on a real industrial application.

Post-processing techniques for viewing and comparing erosion and cyclone efficiency were shown.

