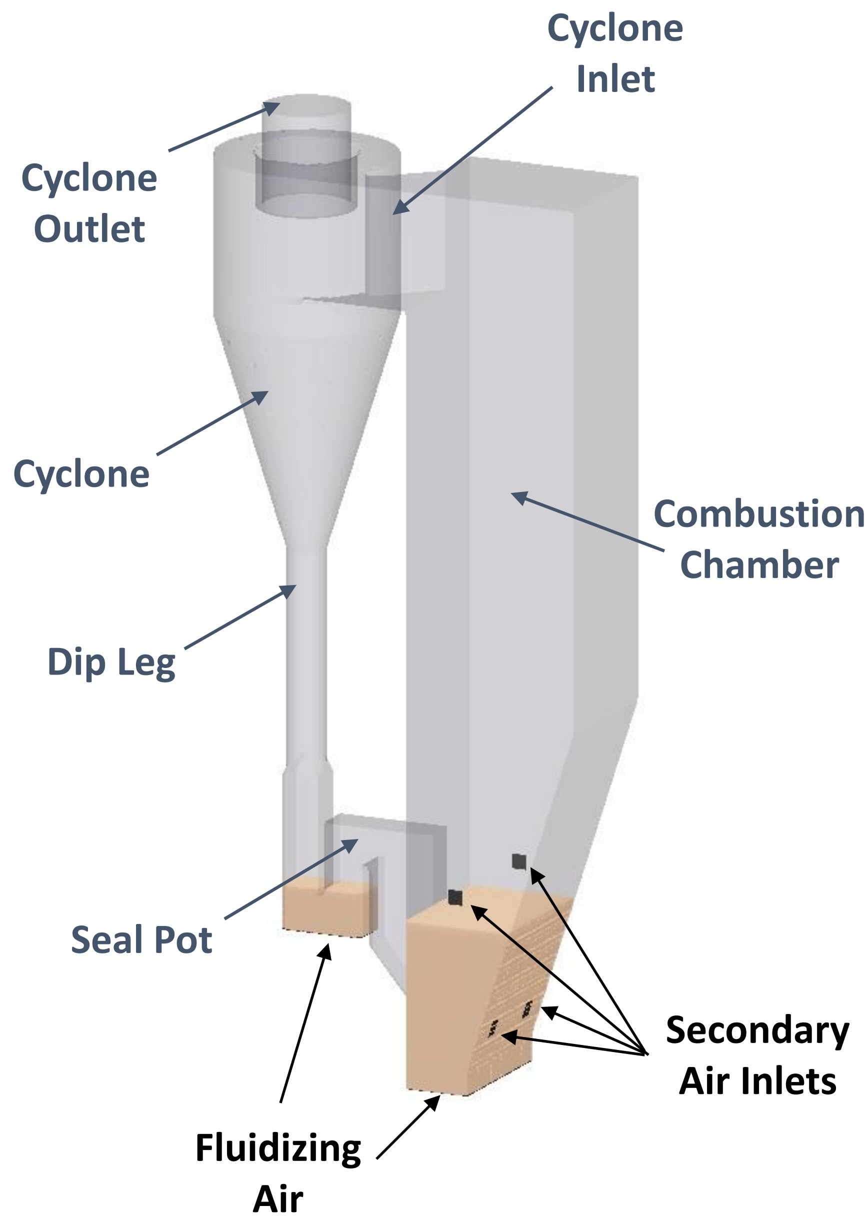
CFB Combustor Erosion Reduction Training Problem

February 2018

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

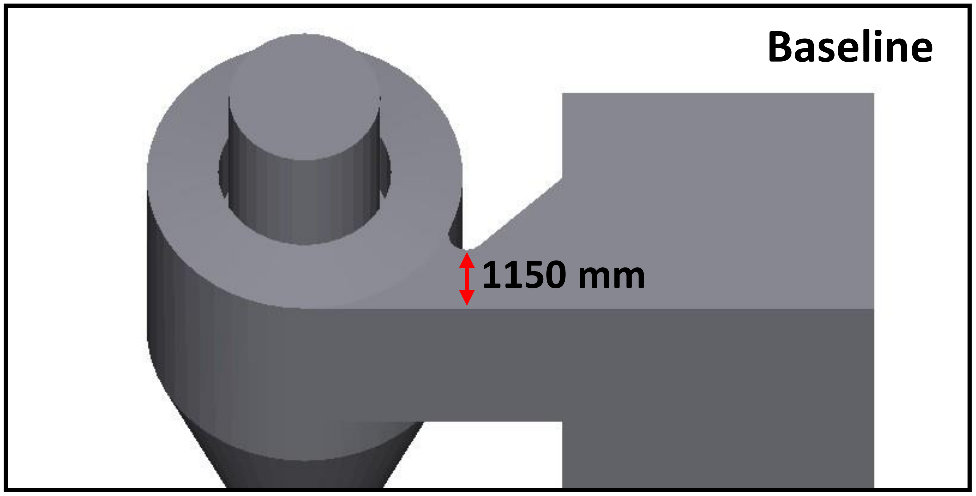
CFB Combustor

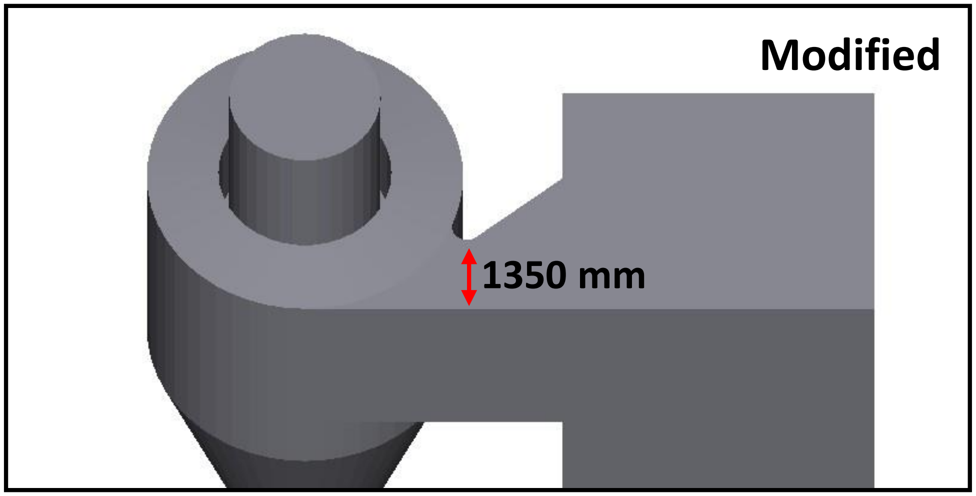
- Model of circulating fluidized bed (CFB) for biomass combustion operating at the 40 MW Strongoli power plant in Italy
- Model consists of combustion chamber, cyclone, and seal pot
- CFB is initialized with 70,000 kg of sand in reactor bed and seal pot
- Model is used to study erosion of internal structures during operation



Baseline vs. Modified Geometry

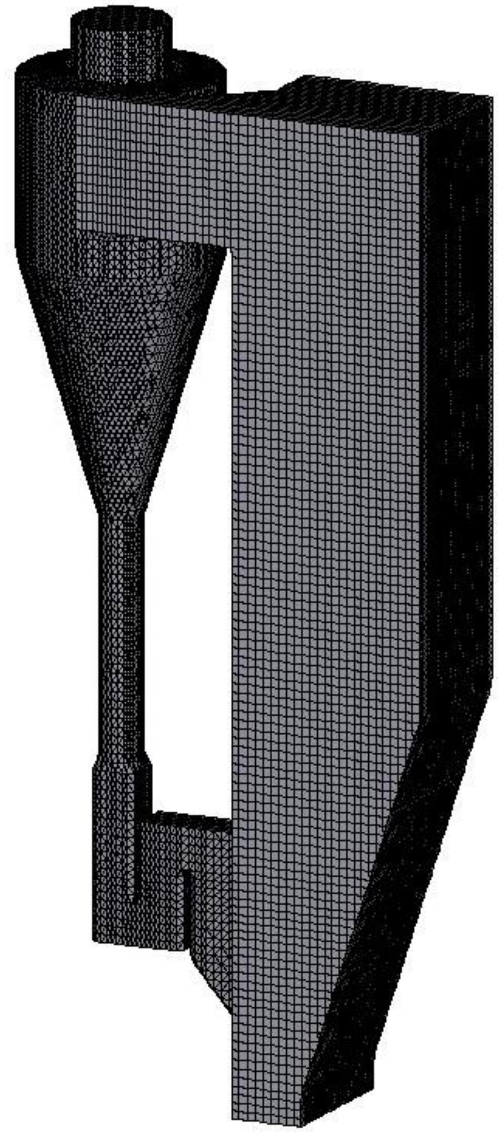
- Training directory contains folders for a baseline and a modified case
- The cyclone inlet width was increased from 1150 mm to 1350 mm for the modified geometry
- It is suggested that trainees start with the baseline case and simulate the modified case as time permits





Grid Generation

- Simplified geometry provided
- Create a grid of about 250,000 real and null cells
- Consider the importance of smooth walls within the cyclone
- Show the instructor your grid before proceeding



System Diagram

Materials Required

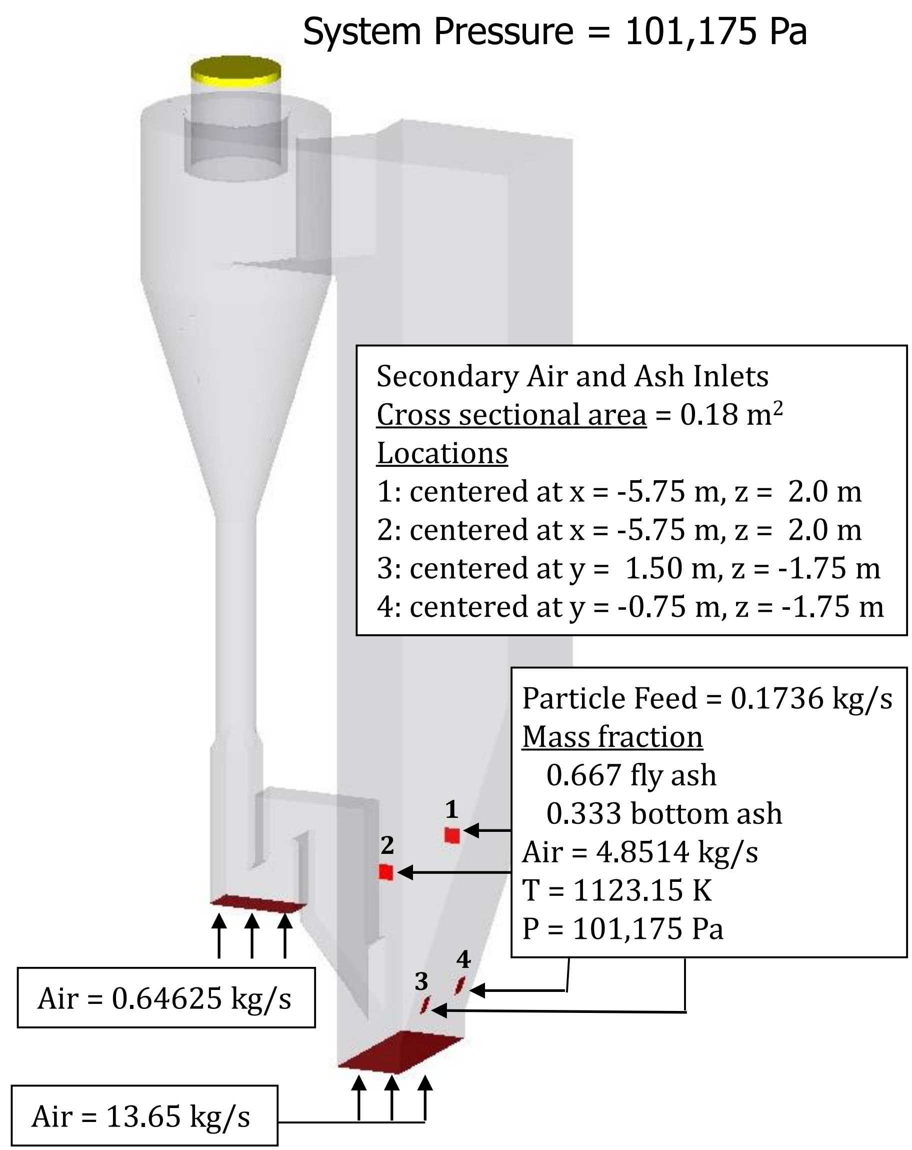
- Air, SiO_2, and Ash

Particle Species

- Sand (SiO_2): ρ = 2650 kg/m^3 and dp = 60 – 480 µm
- Fly ash: ρ = 1500 kg/m^3 and dp = 10 – 480 µm
- Bottom ash: ρ = 1500 kg/m^3 and dp = 18 – 2200 µm
- Full PSD for each provided
- Close pack volume fraction, Θ_{cp} = 0.55

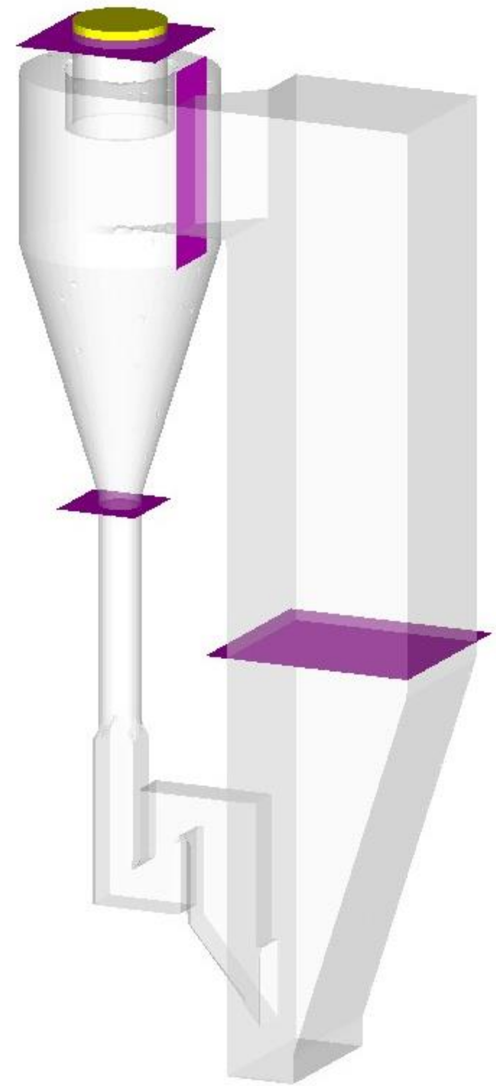
Model Assumptions

- Isothermal flow at 1123.15 K
- A simulation run time of 40 seconds should be sufficient to study erosion
- Particles can exit through the cyclone outlet



Project Setup – Flux Planes

- Define additional flux planes to track flow:
  - At the cyclone outlet
  - At the cyclone inlet
  - Out the dip leg
  - Above the bed in combustion chamber
- Use the **Subdivide by radius** feature to enable fractional efficiency calculation



Project Setup – GMV Output Options

General Mesh View Data Output Options

Please select Eulerian and Lagrangian data for export to the General Mesh Viewer (GMV). Only data selected here can be viewed during post-processing. Values inside parenthesis are the **field names** of the variables within GMV.

Output file interval

Plot interval: 0.1 s   Number of files produced using current end time of **40s**: 401

Eulerian (Cell) Output Data

☑ Particle volume fraction (p-volFra)	☐ Particle bulk density (p-dens)	☐ dp/dx (dp/dx)
☑ Fluid velocity (U, V, W)	☐ Turbulent viscosity (ViscTurb)	☐ dp/dy (dp/dy)
☑ Particle velocity (P_[xyz]Vel)	☑ CFL (CFL)	☑ dp/dz (dp/dz)
☑ Pressure (Pressure)	☐ Particle species (Species)	☑ Particle mass flux (P_[xyz]Mass)
☐ Dynamic pressure (DynPres)	☐ Fluid temperature (f-Temp)	☑ Fluid mass flux (F_[xyz]Mass)
☑ Fluid density (f-dens)	☐ Particle temperature (p-Temp)	☐ Wall heat transfer (wallHeat)
☑ Cell indices (i, j, k)	☐ Cell volume (cellVol)	

Lagrangian (Particle) Output Data

☑ Particle volume fraction (VolFrac)	☐ Particle material (Material)	☑ Velocity (vel[xyz])
☑ Particle speed (Speed)	☑ Particle density (Density)	☑ Residence time (ResTime)
☑ Particle radius in microns (rad)	☑ Particle species (Species)	☐ Residence time by species (ResTime##)
☐ Constant color (Particle)	☑ Unique particle ID (pid)	☐ Temperature (Temperat)
☐ Drag (drag)	☐ Liquid fraction total (liqFrac)	☐ Liquid mass total (liqMass)
☑ Cloud mass (cldMass)	☑ Particles per cloud (npCloud)	☐ Particle mass (mass)

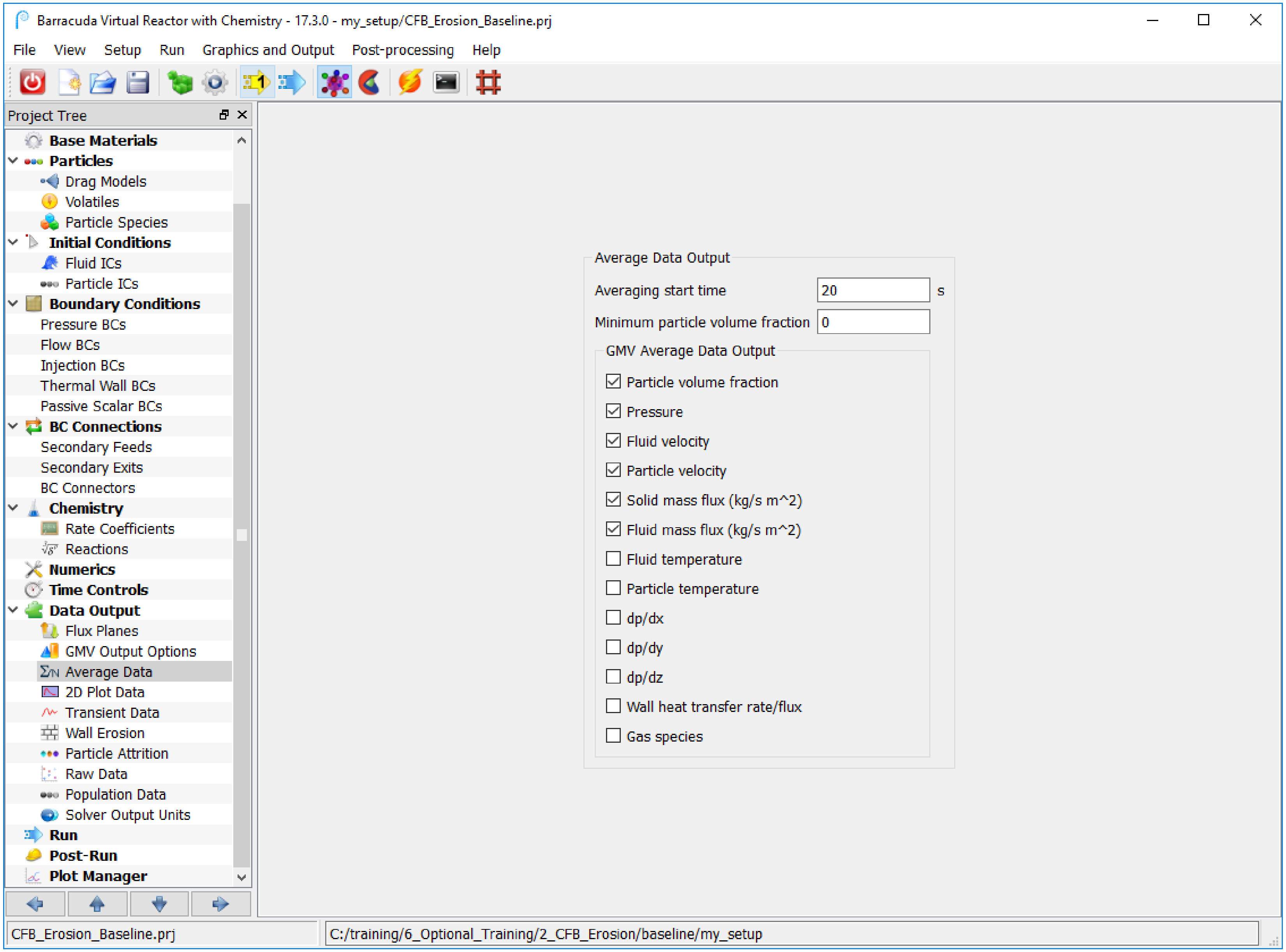
Gas Species

- ○ Mass fraction (<species>.mf)
- ○ Mole fraction (<species>.nf)
- ◉ Mass concentration (<species>.mc)
- ○ Mole concentration (<species>.nc)

Options

- ☐ Compress graphics output (not common)
- ☐ Generate predefined GMV attribute files

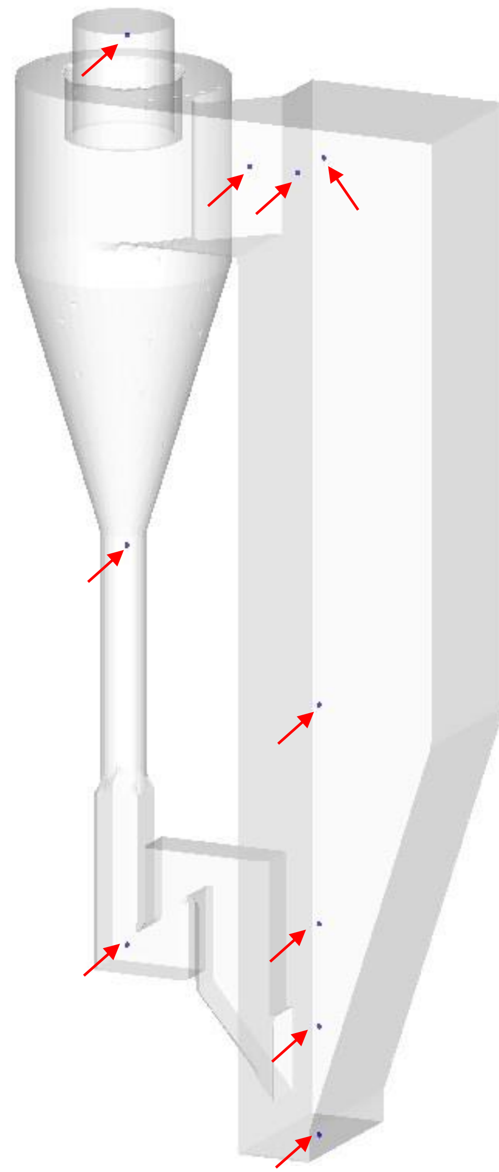
Project Setup – Average Data



Project Setup – Transient Data

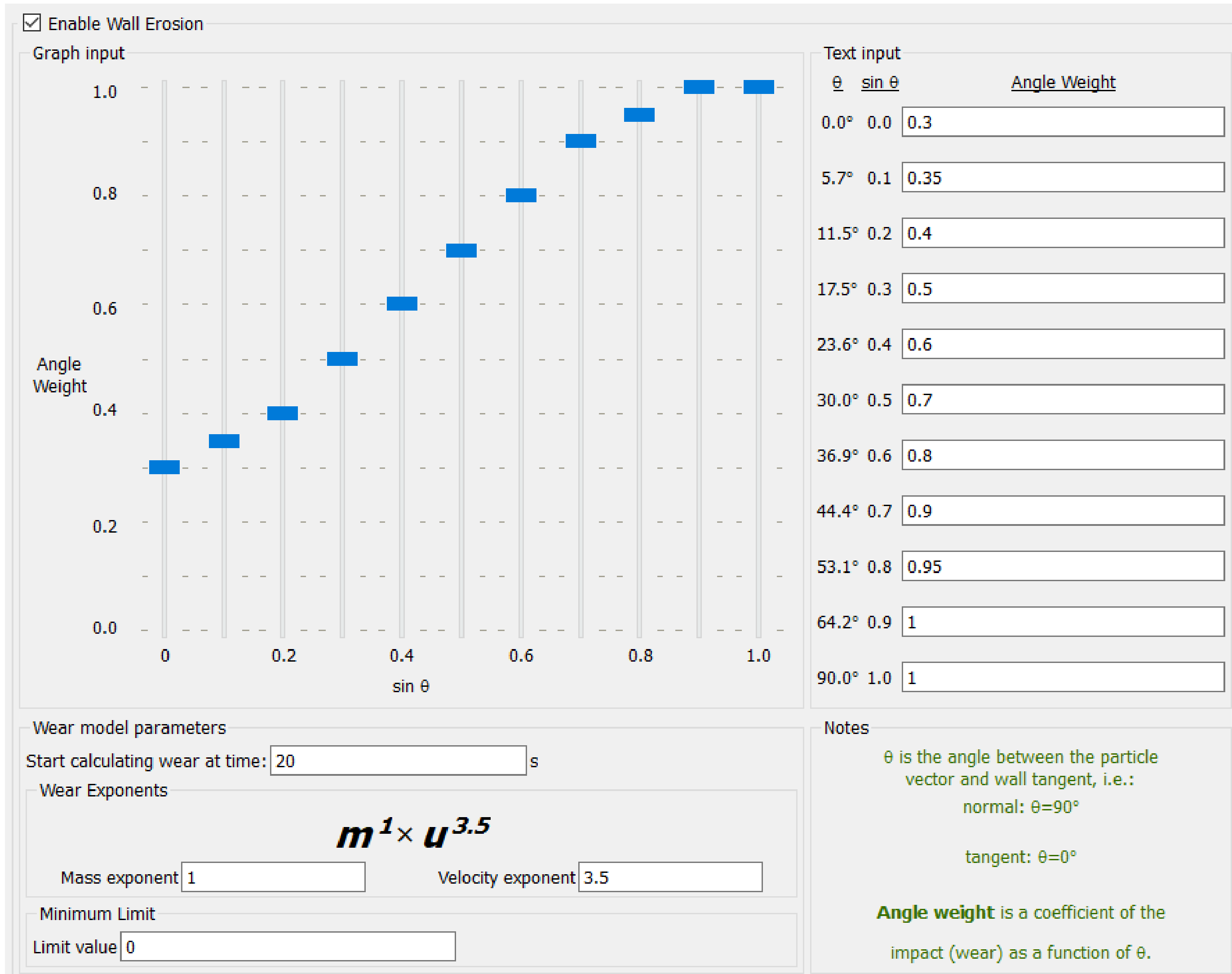
- Select the desired transient data
- Transient data for pressure at these locations is suggested:

x (m)	y (m)	z (m)
-5.471	0.49	-3.461
-5.471	0.49	-0.8
-5.471	0.49	1.7
-5.471	0.49	7.2
-5.471	0.29	20.7
-5.4	1.73	20.7
-4.08	1.73	20.7
0	0	23.1
0	0	10.4
0	0	0.5



Project Setup – Wear Model

- Enable the **Wall Erosion model** and set the angular dependence for refractory (as shown on the right)



Project Setup – Modified Case

- It should be possible to run the baseline and modified cases simultaneously during training
- Once the baseline case simulation is running, set up the simulation for the modified case
- The modified geometry project directory can be created from the existing project directory
- Only minor modifications should be required to change the stl file and re-generate the grid

Creating a New Project from an Existing Project

- To copy the contents of an existing directory to a new directory from the command line use:

  ```
  cp existing_directory/* new_directory
  ```

- If the existing directory contains many GMV files or IC files that will need to be deleted, these can the excluded from the copy command by typing:

  ```
  cp existing_directory/[^G^I]* new_directory
  ```

  - This will exclude any file in the existing directory that starts with G or I

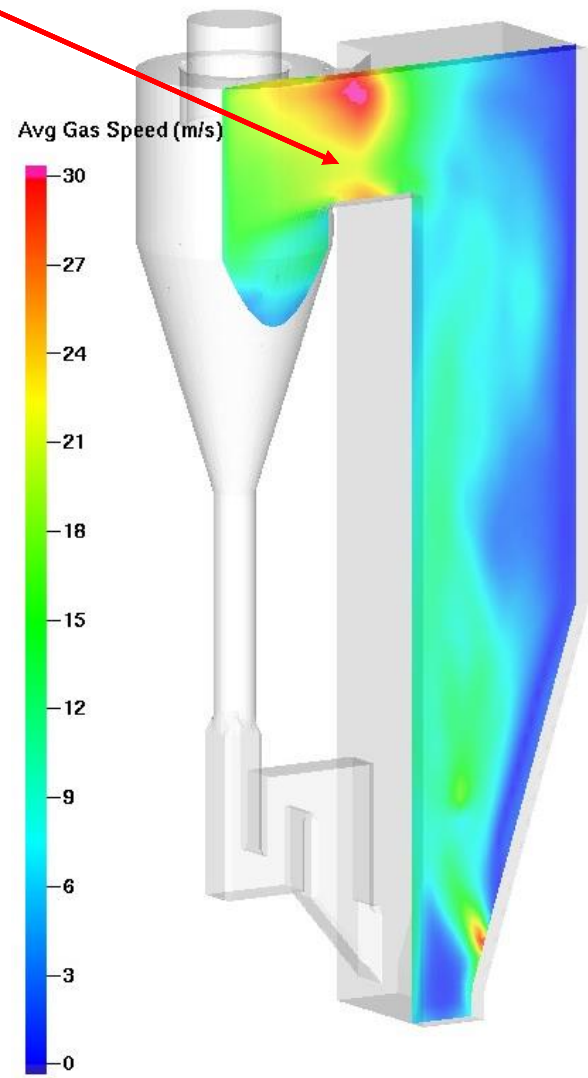
- Note: any other files that meet this criteria will not be copied and will need to be copied separately, for example: `Inlet_velocity.i`

Post-Processing Considerations: Cyclone Erosion

- What are the areas of highest erosion?
- Create a plot of regions most prone to wear
- Hint: Create an **Isovolume** of the **Impact** field and set the limits to isolate the areas of highest erosion
- How does the erosion compare between the baseline and modified geometries?

Post-Processing Considerations: Time Averaged Gas Speed

- Create a plot of time-averaged gas speed [m/s] shown on a plane through the center of the cyclone inlet
- Note: Cutplanes can be created in GMV under the **Calculate** menu
- Does gas speed influence erosion?
- How does the geometry affect erosion?

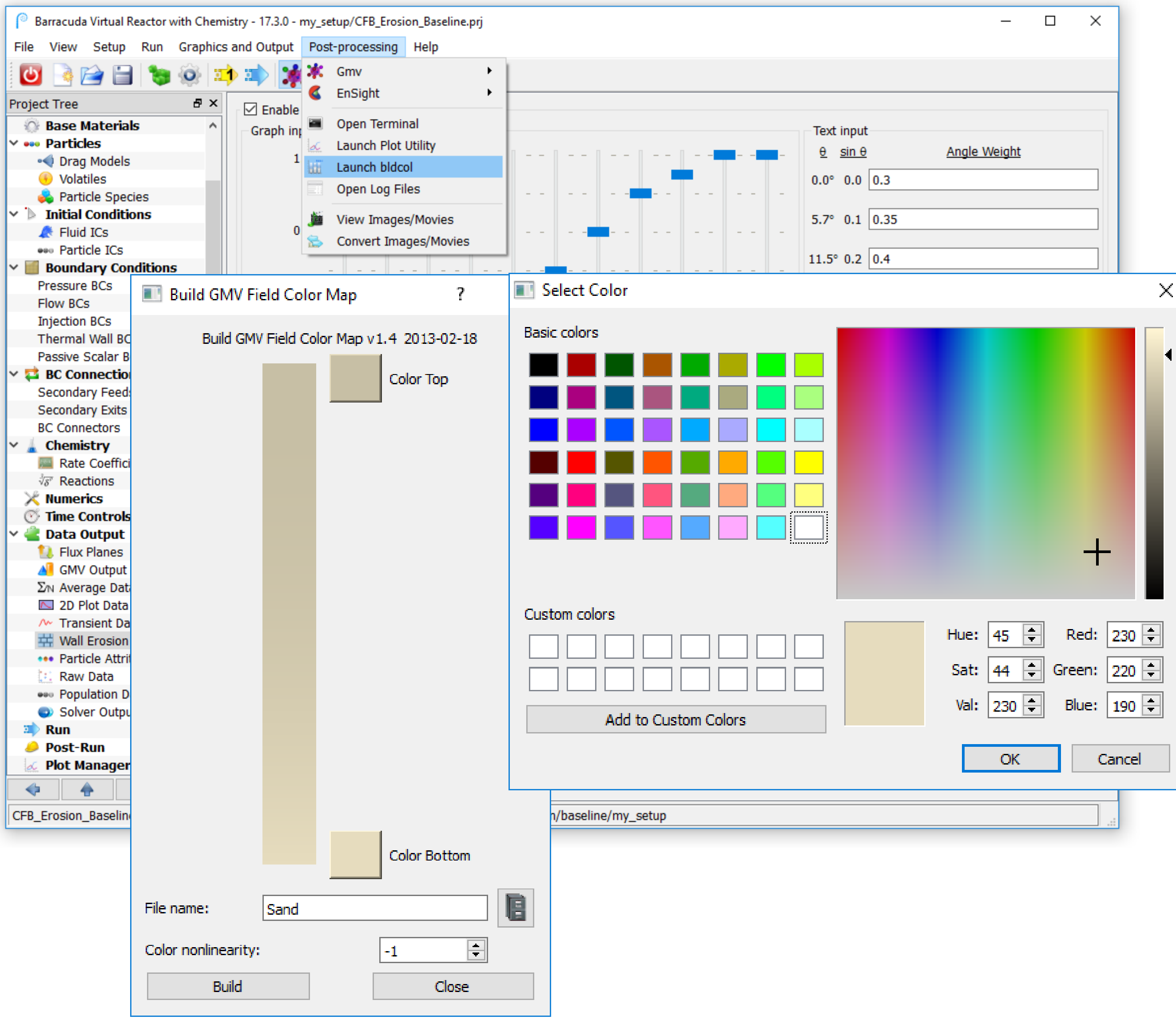


Post-Processing Considerations: Overall Particle Flow Animation

- Create an animation to show an external view of the overall particle flow behavior in the CFB loop
- To do this we'll create a custom colomap for the sand particles and use the BATCHMOVIE script to create the animation
- Details are provided in the next slides

Post-Processing Considerations: Making a Custom Colormap File

- Create a custom colormap for the sand particles
- Under **Post-Processing**, click on **Launch bldcol** and select the desired colors for the particles. Give the colormap file a name such as “Sand” and click **Build**
- In GMV, click on **Ctl-1 → Coloredit → Field Data Color map** and select **Read gmv colormap file**
- Select the desired file and click **OK**



Post-Processing Considerations: Using BATCHMOVIE.sh

- Save the attribute file with a name such as “sand_view.attr”
- Open the terminal for the next step in creating an animation from the attribute file just saved.
- Run BATCHMOVIE.sh with the following command:

```
BATCHMOVIE.sh Gmv sand_view.attr
```

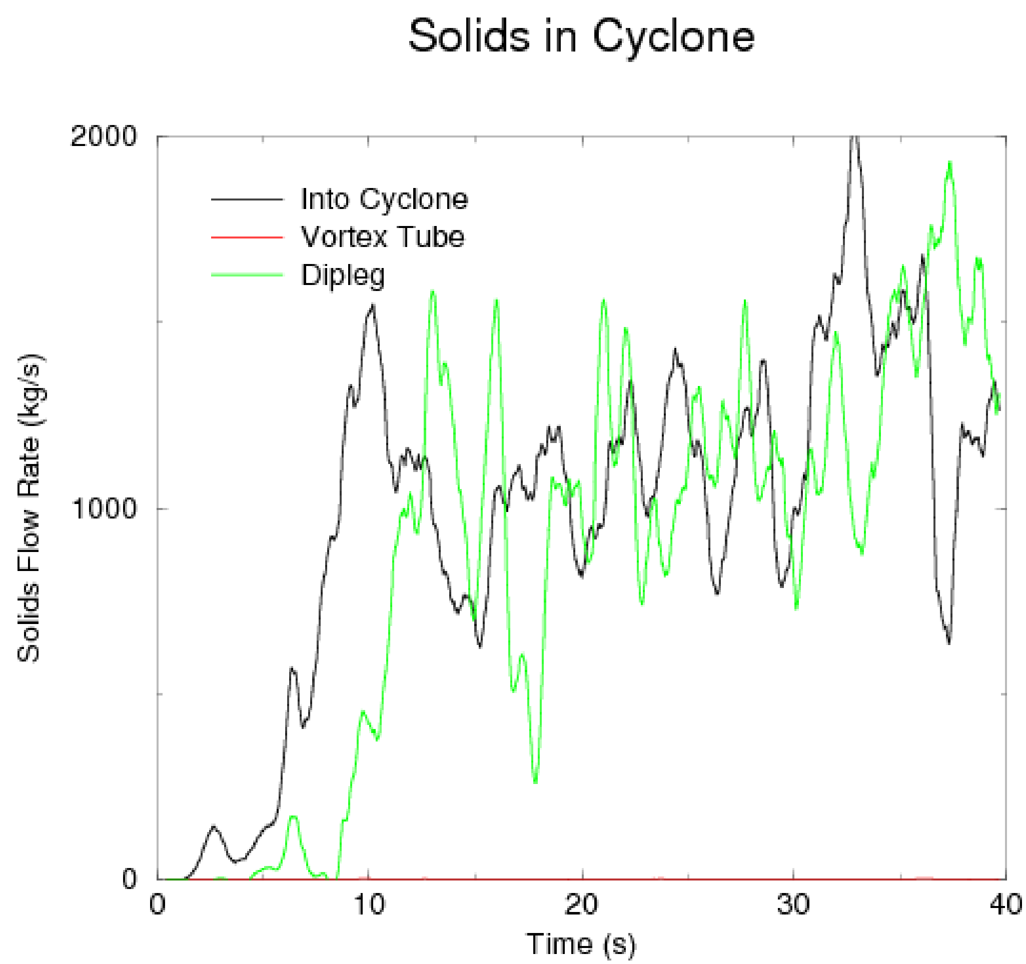
Attribute file name.

GMV files to use – any that start with this pattern will be used.

- You will see information scrolling in the terminal as each Gmv file in the directory is read, and an image is made based on the attribute file specified.
- At the end of the process, BATCHMOVIE.sh will create an animation with name <attribute>.mpg and return control back to the terminal. To play the animation, use the command:
  `xanim sand_view.mpg`

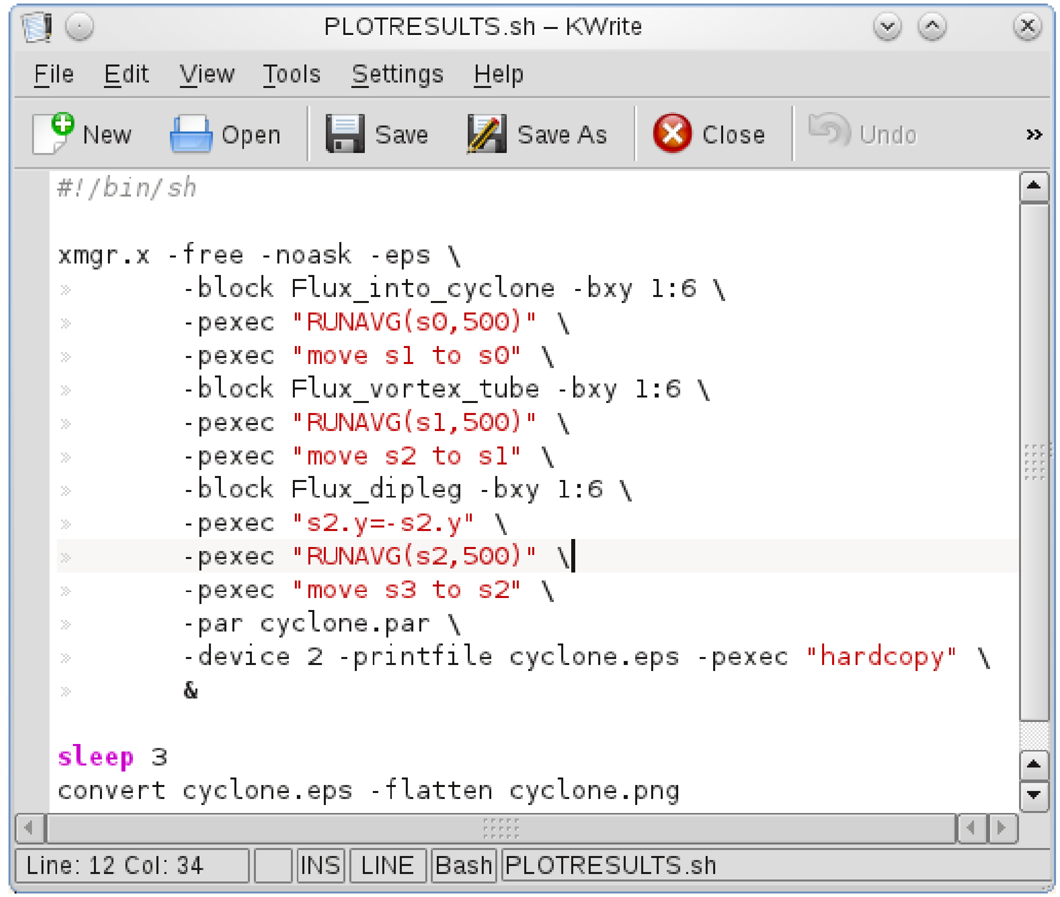
Post-Processing Considerations: Efficiency

- Create a PLOTRESULTS script to look at the flow rate of solids in and out of the cyclone
- Use a running average to smooth out the data
- Example script is shown on the next slide
- What can this tell us about the efficiency of the cyclone?
- How does the efficiency compare between the baseline and modified cases?



Post-Processing Considerations PLOTRESULTS Script

- Refer to the Advanced Analysis section for more information on scripting
- Additional xmgr commands are available at http://plasma-gate.weizmann.ac.il/Xmgr/doc/usage.html#commandline




```
#!/bin/sh

xmgr.x -free -noask -eps \
        -block Flux_into_cyclone -bxy 1:6 \
        -pexec "RUNAVG(s0,500)" \
        -pexec "move s1 to s0" \
        -block Flux_vortex_tube -bxy 1:6 \
        -pexec "RUNAVG(s1,500)" \
        -pexec "move s2 to s1" \
        -block Flux_dipleg -bxy 1:6 \
        -pexec "s2.y=-s2.y" \
        -pexec "RUNAVG(s2,500)" \
        -pexec "move s3 to s2" \
        -par cyclone.par \
        -device 2 -printfile cyclone.eps -pexec "hardcopy" \
        &

sleep 3
convert cyclone.eps -flatten cyclone.png
```

Training Problem Summary

- This training problem showed an example of Barracuda simulations performed on a real industrial application
- Post-processing techniques for building a colormap, viewing erosion, and studying cyclone efficiency were used
- Basic script commands to generate animations using BATCHMOVIE.sh and to create a plot results script for xmgr were presented
- This model was simplified to run during the class. In the real case additional details could be included such as:
  - Details of secondary air nozzles (would require finer mesh)
  - Outlet duct and suspension tube details (would require additional grid)
  - Thermal behavior, chemical reactions, etc. (not likely needed for the erosion study)