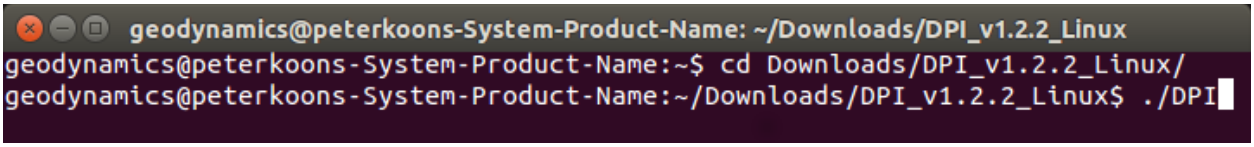
DualSPHysics User Guide for UMaine Numerical Modeling Laboratory Collaborators

The following user guide refers to use of DualSPHysics on the "Everest" computer in the UMaine Numerical Modeling Laboratory.

Full documentation for DualSPHysics can be found in the "DualSPHysics Documentation" folder of the desktop. This includes documentation for the pre-processing interface (DPI) as well as the overall guide.

Use of DualSPHysics can be broken down into 3 basic steps:
1.) Create geometry through pre-processing interface (DPI)
2.) Run a Bash script to find a solution
3.) Visualize output through ParaView software

1.) Create geometry through pre-processing interface (DPI)

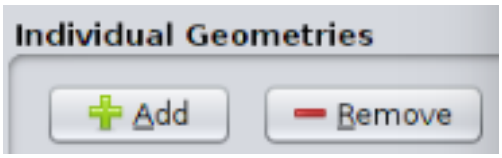


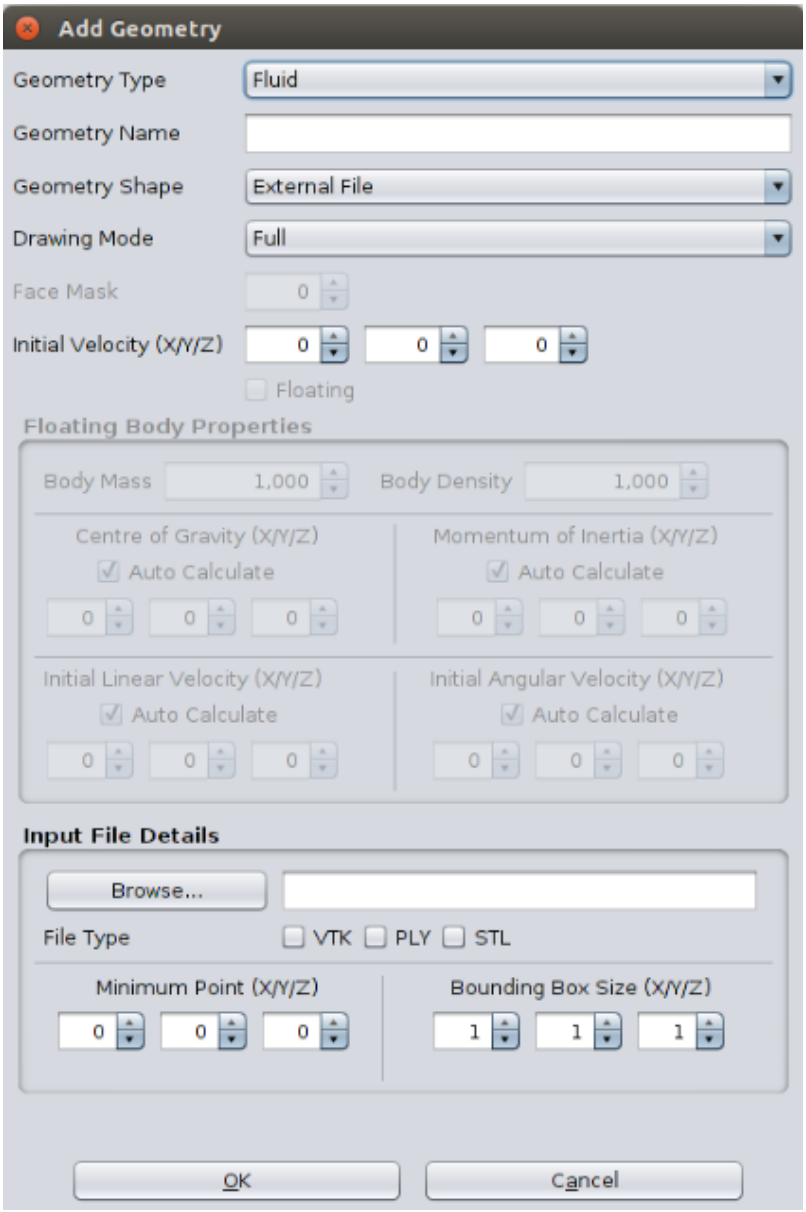
- Open **Terminal**
- Navigate to Downloads/DPI_v1.2.2_Linux to **run DPI** by typing the following into terminal:

  cd Downloads/DPI_v1.2.2_Linux
  ./DPI
- in DPI, navigate to "**Contants**" tab
- specify number of boundary particles and fluid particles

> NOTE: Look at example cases for ballpark figures for values of constants
> - Downloads --> DualSPHysics_v3.1_Linux_x64 --> RUN_DIRECTORY
> - CASEDAMBREAK is a good example
>   - CaseDambreak_Def.xml using gedit (open with --> gedit)
>   - Constants start on line 4 of code
> - constants can always be changed in the code later on

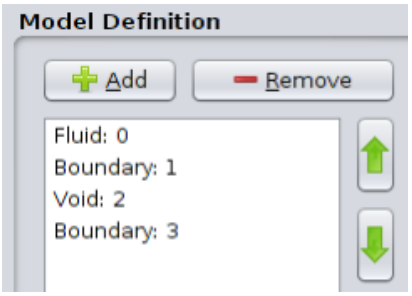
- in DPI, navigate to "**Geometry**" tab
- specify global properties
- look at example cases for ballpark figures
- start with 0 for X/Y/Z of Minimum Boundary Point (line 18, "<pointmin" in CaseDambreak.xml)
- Boundary Size is the size of the interface (line 19, "<pointmax" in CaseDambreak.xml)
- Distance Between Particles dictates how densely-populated particle field will be when solving. Start with larger number to begin with (about 0.3) in order to ensure correct geometry, then reduce DBP from there. This effectively controls resolution.
- Drawing Output Options --> DP gives movement





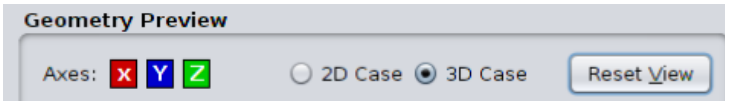
(critical), Boundaries (critical), others are less critical

- Individual Geometries
- To create a geometry, click the "add" button
- Geometry Type, Name, and Shape can be determined in the window shown above
- Geometry Shape can be from an imported file (allowable file types are listed in the DPI user guide. See DPI-V1.2_Guide.pdf under "downloads")
- Box Properties are at bottom
- Check "Floating" if you want to allow the object to float



NOTE: Numbering individual geometries is wise because within the code, each geometry is going to have a make number.

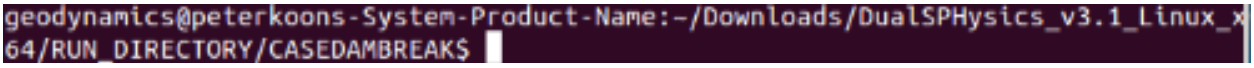
- Once individual geometries are numbered, add geometries under "Model Definition"



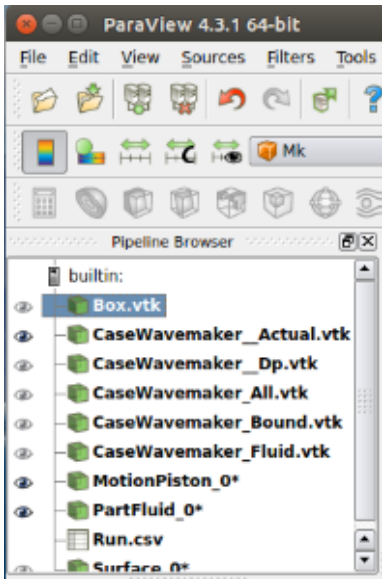
- Select solution for either 2D or 3D case
- In DPI, navigate to "**Parameters**" tab
- look at example cases for ballpark figures
- Before saving, create folder name (such as "CASEEXAMPLE") in RUN_DIRECTORY
- File --> Save As... --> (CaseExample_Def.xml)
- Without the _Def.xml (which is case sensitive), the file will not be read

2.) Run a BASH script that solves for solution

- File --> Copy example case GPU script (such as CaseWavemaker_linux64_GPU.sh) into new project folder, then rename to match new project name
- Open BASH script
- Change name: name=CaseExample
- Executable files are called from directory.
- Boundaryvtk: see DualSPHysics guide for additional commands that can be added on
- Partvtk: see DualSPHysics guide for additional commands that can be added on
- File --> Save
- After creating XML file, open it in gedit and compare it to example file to ensure that parameters have been properly translated to new file (the struggle is real).
- Boundaryvtk designates output files and is where the make number comes into play.
- Navigate to project folder within RunDirectory (you should see Bash script here)

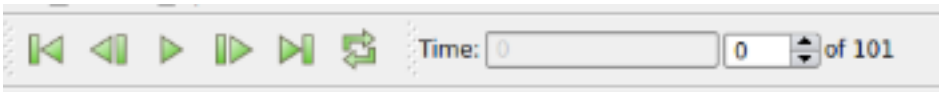


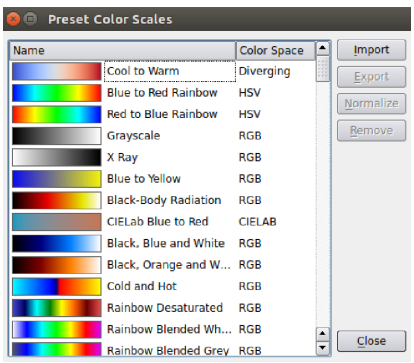
- Bash script needs permissions to be run by all users
- chmod 755 filename
- ./CaseExample
- Minutes to hours of processing time are to be expected and largely depend on dp value and length of simulation



3.) Visualize output through ParaView software

- Home --> Paraview --> bin --> paraview
- File --> Open --> CaseExample --> Select all files within output folder
- vtk files are on left hand side in pipeline browser
- The files with asterisks have multiple files
- Paraview will automatically take timestep files and put them in order





- Press Play to see visualization of solution.
- Time: you are able navigate through timesteps manually
- Fluid --> Id box change to Vel (Velocity)
- Change boxes to right to visualize magnitude, x, y, z, surface
- Color Scheme: Color Map Editor is on right side of ParaView window
- Click "choose preset" icon (third from bottom)