Questions & Answers

WEBINAR: What's New in *FLAC3D* v9 (May 9, 2023)

How can we determine Rayleigh parameters (Damping ratio and central frequency)?

- Rayleigh damping is accurate at two controlling frequencies. For example, at omega_1 and omega_2, both having damping ratio D, we can determine D using the following equations: D = 0.5 * (alpha / omega_1 + beta * omega_1) and D = 0.5 * (alpha / omega_2 + beta * omega_2). By solving for alpha and beta, we can find the values for these parameters.

- The center frequency can be calculated as (alpha / beta)^0.5, and the minimum damping ratio is given by (alpha * beta)^0.5. It's important to note that the actual damping will be less than D within the range of omega_1 and omega_2, and greater than D outside of this range. However, this is the characteristic of Rayleigh damping.

- The selection of omega_1 and omega_2 is up to the user and is typically related to the natural frequency of the model. For instance, omega_1 can be set as the natural frequency of the model, while omega_2 can be chosen as 3 or 5 times omega_1. It is essential that the range of omega_1 and omega_2 covers the dominant frequency of the input motion.

Can we interpolate material properties e.g. modulus within *FLAC3D* model if modulus values are known at few discrete points?

- This can be achieved by implementing a *FISH* or Python subroutine.
- Here's a Python example for importing pore pressures:
  - https://www.itascacg.com/learning/tutorials/python-and-pore-pressure-initialization

Do you have any plan to improve boundary conditions to represent half-space in static problems?

- For periodic boundary, we can manually use a *FISH* function to attach gridpoints at two boundary sides.
- For a pure half-space in static problem, we would need more details.
  - Please contact *FLAC3D* Technical Support: https://www.itascacg.com/contact-itasca/sw-tech-support-flac3d.

Is the plastic Damage concrete useful to simulate a concrete beam or a concrete bulkhead/barricade under load?

- We foresee no impossibility.

Can you import multiple polylines as centerlines of tunnels and extrude the tunnel profile ? This would be useful to simulate multiple tunnels in a mine for example.

- Yes, we demonstrated a manually created tunnel along a pre-defined curve.
- However, intersecting tunnels is not currently, easily possible via Sketch. For this, we would recommend using *Griddle*.

Can intersecting zone joints be build only with CAD and *FLAC3D* or do we have to go with *Griddle*?

- *FLAC3D* v9 does not include the same cutting logic that in *3DEC*. The cutting must be built with CAD and *FLAC3D* or *Griddle*.
- Like with interfaces, you can define a Zone Joint between groups of zones.

Could I ask you, why the developer team changed the syntax commands version by version?

- There are very few syntax changes from v7 to v9.
- We renamed the "extruder" panel to "sketch" to avoid confusion because *FLAC2D* v9 now shares the same framework and the term "extruder" is less meaningful in the context of 2D. However, data files that use the “extruder” command will still work.
- The addition of new features or capabilities may also require new commands and keywords.

Could you please share with us the model of the tailings dam that you have created with the Norsand model? I mean the data file. Thank you.

- Yes, we will send this datafile to your email shortly.

How can a landslide-induced pressure on a tunnel be modelled the better in this version?

- We would need to know what the limiting issue was for the modeling of landslide-induced pressure on a tunnel in the old version.
- Please contact *FLAC3D* Technical Support (https://www.itascacg.com/contact-itasca/sw-tech-support-flac3d) or post your question to our Software Forum (https://forum.itascainternational.com).

(1) Can we apply cyclic loading on the structure? (2) Do we have option for user define contact model or option for the change in the existing contact model?

1. Yes.

2. Assuming you are referring to Zone Joints, yes you can use any built-in *PFC3D* contact model and with the *PFC3D*, and its UDM option, create custom contact models.

You showed how to create a curved tunnel. How can we create the host rock around this tunnel?

- Automating this capability is a work in progress.
- Currently, there a couple of options:
  1. Mesh and extrude the tunnel using a structured mesh and then use building-blocks to build out the surrounding mesh.
  2. Add a boundary around the tunnel and include this in the extrusion. To add a 3D topography, use the zone generate from-topography command. We are working on an example for this approach.

(1) Is there a plan to include Maxwell Damping in *FLAC2D*? (2) There was a section in the outline regarding "Zone Joints", will that still be covered today?

1. Maxwell damping is already available in *FLAC2D*.
2. Yes, zone joints were covered in the webinar.

(1) Will there be added an option so that interfaces recognize linear elements? (2) Can you increase zone deformability so that one can model impacts or blasts?

1. Currently *FLAC3D* has no direct way to set interfaces between liners. It's theoretically possible by manually setting deformable node-to-node links, but this may be tedious.

2. For large deformations, a better approach is remeshing where poor zone quality is detected. This was available in *FLAC* but is not built-in for *FLAC2D* or *FLAC3D* currently. People have written *FISH* functions to do this for quadrilateral meshes, but unstructured meshes are more difficult. *FLAC3D* is currently used to model blasting (PPV, damage) but for cases where fragmentation and breakage are desired, a DEM or Lattice code such as *3DEC*, *PFC* ,or Blo-Up should be used.

Any in person *FLAC2D* training coming up?

- We just had an in-person *FLAC2D* training course in Minneapolis this past April.
- This course is being processed and will be posted to our Software Academy training site as soon as possible.
- You can also request paid, customized training here: https://www.itascacg.com/learning/custom/customized-software-training.

How is water pressure input into Zone Joints?

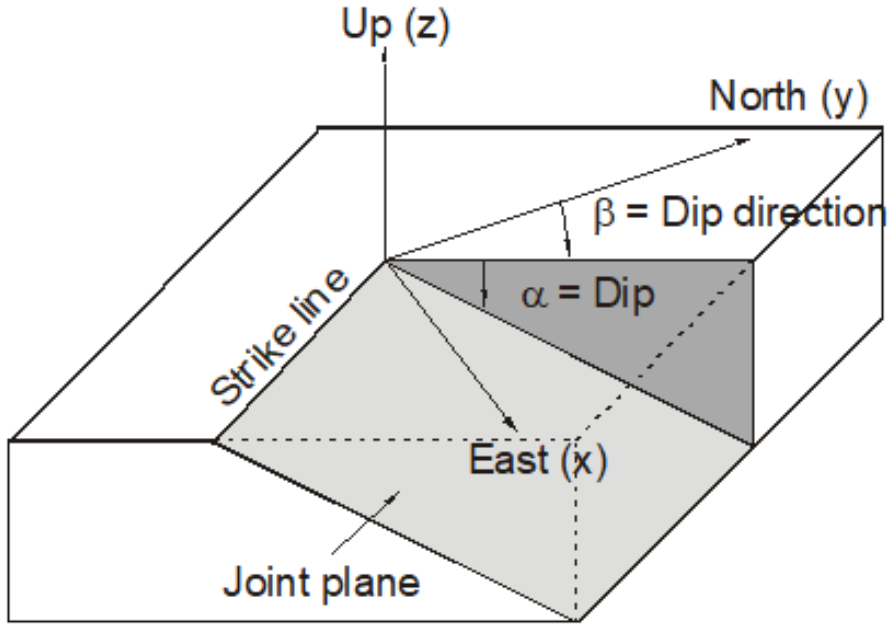
- It is the same as for traditional interfaces.

Can the flow model be used for compressible fluids? How can the new version can be used for CO_2 storage projects?

- Yes, our fluid flow methods model compressible fluid flow (Darcy flow only). They do not model incompressible fluid.
- Suggest a Google search to see what's been published in the literature.

Any document on orientation conventions? I see you have used a dip value > 90 degree.

- In Itasca 3D software:



  - Note that angles are expressed using degrees in commands and using radians in *FISH*.

- In those tunnel figures for Columnar-Basalt, this was just to show the two sets of joints are perpendicular each other.

How can 3D geosynthetic material be modeled (e.g., geosynthetic reinforced earth wall)?

- Geogrid structures are available in *FLAC3D*.
- Advanced geogrids can be modeled using nonlinear cable elements that can yield and rupture if the strain limit is reached. During the strength-reduction analysis, if a cable element reaches the limit tensile strain, it is considered to have ruptured, and its contribution to the model is removed. To learn more, refer to the *Continuum Deformation Analyses* section in the paper "Stability and Stress-Deformation Analyses of Reinforced Slope Failure at Yeager Airport".
- For further assistance, please contact our support team and provide them with additional details and background information. They will be able to provide you with more comprehensive guidance and support. You can reach them at https://www.itascacg.com/contact-itasca/sw-tech-support-flac3d or post your question to our Software Forum at https://forum.itascainternational.com.

Concrete damage plasticity is a welcome addition. What is the degree of mesh dependency and how is it addressed to produce physically meaningful results? Any regularization techniques added? Thanks.

- Yes, this model has two properties called "compression-length-reference and "extension-length-reference". The advanced users can play with these two parameters to reduce mesh dependence.

Any plans to incorporate more general 3D meshing capabilities (e.g., cut brick with fault surface geometry and mesh the two sides and fault surface) inside *FLAC3D* as opposed to outside in *Griddle*?

- In our long-term plan, we intend to incorporate *3DEC* logic for cutting. However, it is unlikely that this feature will be available in version 9.

What is your recommended software other than "Rhino" for creating ".stl" files?

- Any CAD or meshing software that can export an STL file should be fine.

(1) How to use random fields to simulate the uncertainty of soil profiles in *FLAC3D*? (2) How to simulate random fields in a soil profile of a slope in *FLAC3D*?

1. Use Python to interpolate properties.

   - Here's a Python example for importing pore pressures: https://www.itascacg.com/learning/tutorials/python-and-pore-pressure-initialization.

2. Use the command called zone property-distribution, which can input properties in space by Gaussian or uniform distribution.

The dynamic timestep is controlled by the P-wave velocity and the zone dimensions, which I would think impose a fundamental limit on timestep. So how have you managed to improve this by 2.6 times?

- We have optimized the safety factor to have larger stable dynamic time steps.

We have been working with Maxwell damping for a while now and it appears to affect stiffness. Has this been resolved, or do we need to manually adjust the stiffness of the structure/zones where Maxwell damping was applied?

- There is ongoing debate regarding this matter.
- Some experts argue that this effect is dynamic in nature, while others suggest compensating for it by using a smaller stiffness.
- Ultimately, the significance of this effect in a project depends on the user's understanding and interpretation.
- It is essential to consider the specific context and project requirements when determining the appropriate approach to address this effect.
- Please contact *FLAC3D* Technical Support (https://www.itascacg.com/contact-itasca/sw-tech-support-flac3d) or post your question to our Software Forum (https://forum.itascainternational.com).

To perform probabilistic analysis, numerical model is to be run for a number of sample for the input variable. So how to import a table (either in .txt or excel format) which contains all the samples points?

- This can be achieved by implementing a *FISH* or Python function.
- For example, the Python module openpyxl is now included with v9 software
- This is a Python library to read/write Excel 2010 xlsx/xlsm/xltx/xltm files.

Is Python and array computation fully supported for structural elements and structural links in *FLAC3D* v9.0?

- This is an ongoing work.
- We will gradually add and complete Python functions for structures for v9.

Can we change specific colors of the contour legend?

- Indirectly, the answer is yes, since we can specify colormap pattern.