

Prelimina

User Guide

Written for Prelimina 0.3.2
Updated 2 November 2026

PLASMICA d.o.o. · prelimina.com
Prelimina is in development and not yet ready for production use.

Contents

Getting started 3
- Choose how to run Prelimina 3
- Check your computer 3
- Install the desktop application 4
- Run your first simulation 4
- Find your results 7
- Change something and run again 7
- Keep Prelimina up to date 8
- What to try next 8

The workspace 9
- The main window 9
- The three stages 9
- Setup files 10
- Select and edit 10
- View tools 11
- Simulation preview 11
- Explore a setup 12
- Settings menu 13
- Keyboard shortcuts 14

The scene 15
- Build a sloshing tank 15
- The domain 19
- Fluids 20
- Bodies 20
- Initial liquid 21
- Motion 22
- Inlets and outlets 23
- Probes and measurements 24
- Next steps 24

Prepare a run 25
- Correct the tank 25
- Read the panel 26
- Grid cell size 27
- Time stepping 28
- Pressure and transport 30
- Check the memory 30
- Check a setup before a long run 31
- Next steps 31

Run and read the results 32
- Control the run 32
- Choose what to show 32
- Look inside with a slice 34
- Read the diagnostics 34
- Compare with a measurement 34
- Save the numbers 35

Open results in ParaView 36
If a run stops 36
Next steps 37

Getting started

This chapter takes you from nothing to a finished simulation. You will open a bundled sloshing tank, run it, look at the flow, and change one setting. It takes about 15 minutes.

> **Prelimina is in development.** It is useful for exploring ideas, but it is not yet ready for production use. Back up your projects and keep them outside the installation folder.

Choose how to run Prelimina

Prelimina runs as a desktop application or in your browser. The interface is the same in both.

	Browser	Desktop
Install	None. Open prelimina.com/app	Windows or Linux installer
Graphics	WebGPU in Chrome or Edge 137 or later	Vulkan 1.3 GPU with a recent driver
Solvers	FLIP, single phase	FLIP, single phase. FVM, FIRM and the two-phase variants are locked beta features.
Results	View only, no results folder	Saved to disk and exported as VTK
Examples	One dam break	Four, listed below

The browser version is the quickest way to see Prelimina. Use the desktop application for your own work. The rest of this chapter follows the desktop application, and the steps are the same in the browser unless a note says otherwise.

> **In the browser.** Firefox and Safari cannot run Prelimina yet. If Chrome reports WebGPU as off or falls back to a software renderer, open `chrome://gpu` and check that it shows **WebGPU: Hardware accelerated**. On a software renderer the simulation stops during initialisation with an error about storage buffers.

Check your computer

	Windows	Linux
System	Windows 10 22H2 (64-bit) or newer	64-bit x86 Linux, glibc 2.28 or newer
GPU	From 2015 onwards, Vulkan 1.3, recent driver	The same
RAM	8 GB minimum	The same
GPU memory	4 GB for small scenes, 6–8 GB preferred, 8–12 GB or more for larger work	The same
Other		FUSE 2 to mount the AppImage

These are provisional targets while Prelimina is in development. The Download section always has the current list. macOS is not available yet.

Install the desktop application

Windows

1. Download the Windows installer from the Download section.
2. Run `Prelimina-0.3.2-win-x64-Setup.exe`. The installer is not code-signed yet, so Windows may warn you before it runs.
3. Start **Prelimina**.

Linux

1. Download the AppImage from the Download section.
2. Keep it in a folder you can write to, so that it can update itself.
3. Make it executable and start it:

```
chmod +x Prelimina-0.3.2-linux-x64.AppImage
./Prelimina-0.3.2-linux-x64.AppImage
```

If your system has no FUSE 2, start it with `APPIMAGE_EXTRACT_AND_RUN=1` set in the environment.

> **Licence.** Prelimina is free for noncommercial use. Commercial use, including business evaluation, needs a paid licence before the work begins. See the licence agreement.

Run your first simulation

Prelimina organises work into three stages, shown as tabs at the top of the window:

- **Scene** holds the physical problem: geometry, fluids, initial state, motion and duration.
- **Prepare** holds the numerical choices: solver, grid size and time-step controls.
- **Simulate** runs the solver and shows the results.

When you start Prelimina it opens an empty, untitled scene in the **Scene** stage.

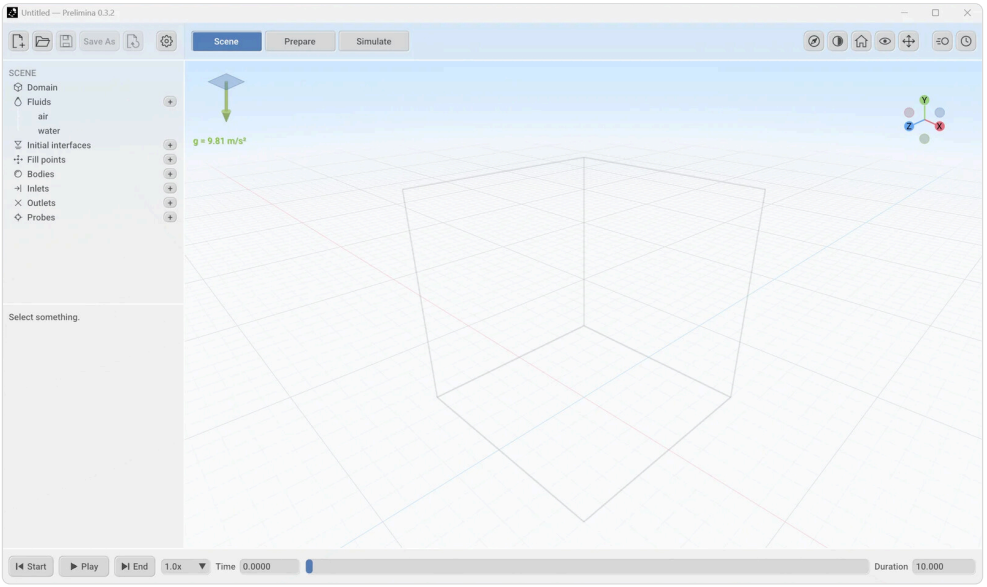


Figure 1. Prelimina on first launch. The Scene stage is open, with an empty domain.

Open the example

1. Click **Open**, the folder icon at the top left, or press `Ctrl+O`. On a new installation the dialog starts in the examples folder.
2. Open the `sloshing_tank` folder and choose `sloshing_tank.json`.

Prelimina copies the example into `Documents/Prelimina/Examples` and opens the copy. Your edits and results stay there, and the original stays unchanged. Opening the same example again makes a new numbered copy.

> **Can't find the examples?** Windows may open the dialog in the last folder you used instead. The examples are in the `examples` folder beside the Prelimina program, by default `%LocalAppData%\Prelimina\current\examples`.

The tank appears in the viewport. It is 1 m long, 0.6 m high and 0.1 m wide, and closed on all sides. The liquid starts about 0.3 m deep with its surface tilted by 10°. When the simulation starts, gravity pulls that surface flat and the liquid sloshes from side to side.

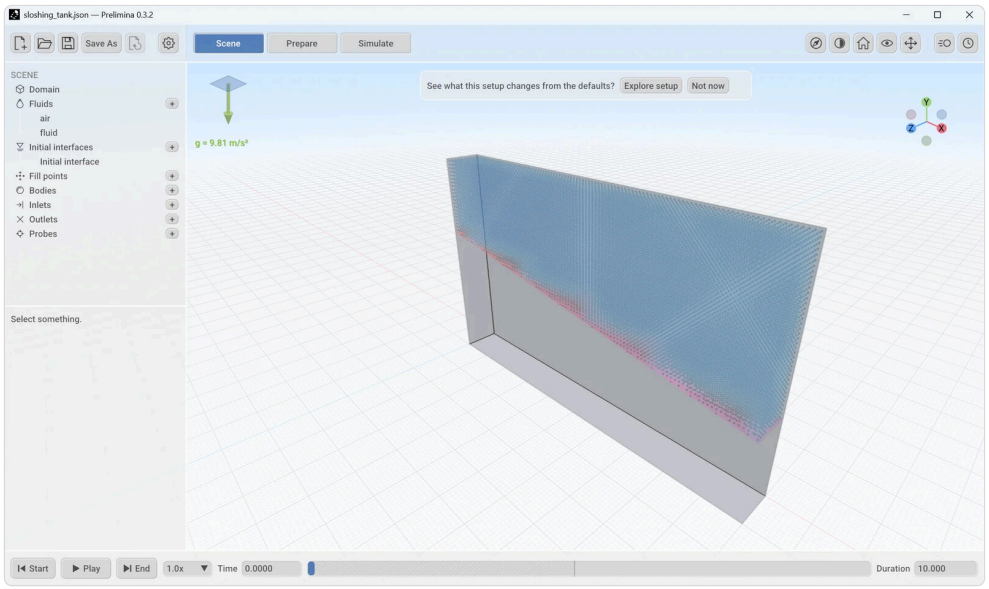


Figure 2. The example open in Scene. The blue volume is the liquid, tilted at the start of the simulation.

A banner offers to show what this setup changes from the defaults. Click **Not now** for this first run. You can start the tour later with **Explore setup** in the settings menu.

Look at the settings

Click **Prepare**. This example uses the **FLIP** solver with the **Single phase + void** variant, which simulates the liquid and treats the space above it as empty. **Grid cell size** is 0.02 m.

Below the viewport, Prelimina predicts the peak memory for the run and compares it with what your GPU has available. Check this line before you start a large run.

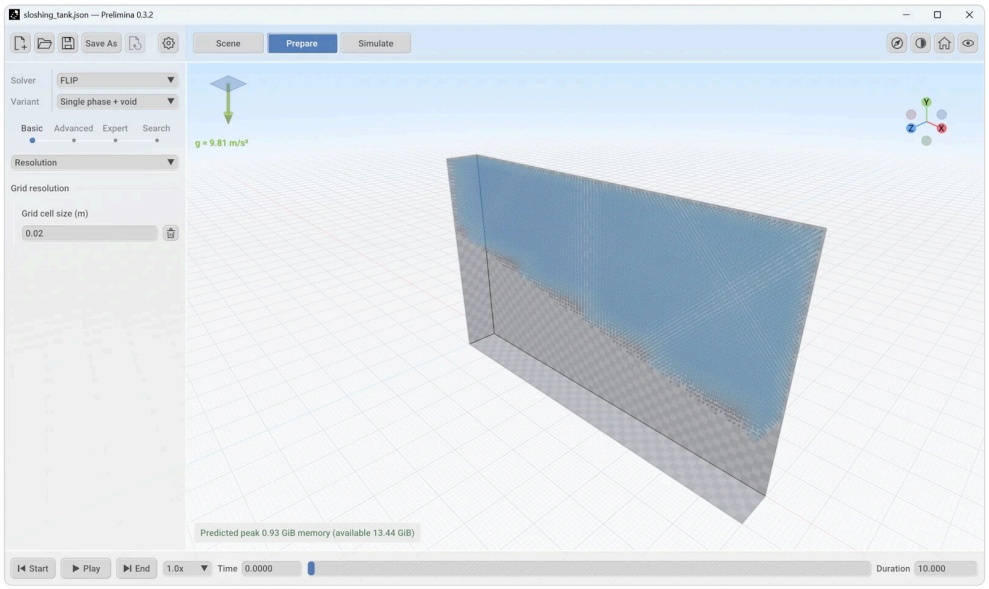


Figure 3. The Prepare stage with the solver, grid cell size and predicted peak memory.

Run it

1. Click **Simulate**. The panel shows **Simulation not initialized**, because nothing has been sent to the GPU yet.
2. Click **Run** at the bottom left.

The first run on a computer shows **Optimizing the simulation for your device**. This can take a minute, and later runs start faster. In the browser the first visit can take several minutes.

The liquid then starts to move, and the bar at the bottom shows the simulated time. This example simulates 10 s, which took a little over a minute on an NVIDIA RTX 5080.

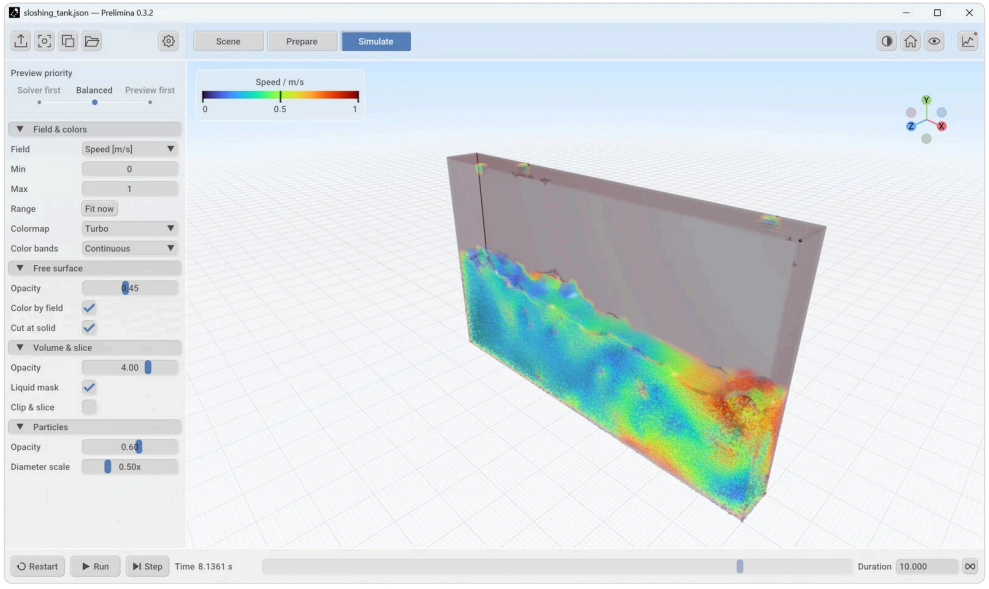


Figure 4. The running simulation, coloured by speed.

Press `Space` or click **Pause** to stop and continue. **Step** advances one time step while paused. When the bar shows **Simulation finished**, click **Restart** to run again from the start.

Look around

These controls work in every stage while the pointer is over the viewport:

Action	Control
Orbit	Left-drag
Pan	Right-drag
Zoom	Mouse wheel or middle-drag
Focus on a point	Double-click it
Show everything	**Frame all**, the house icon at the top right
Switch between perspective and parallel view	`0`

The small axis gizmo at the top right shows the orientation. Click an axis to look along it.

Read the results

The panel on the left controls what you see. **Field** selects the quantity. **Min** and **Max** set the colour range, and **Fit now** fits the range to the current values. In this example the speed goes above the default maximum of 1 m/s, so the fastest flow shows as solid dark red until you change the range.

Click the graph icon at the top right to open the live diagnostics. **Infos** shows the size and speed of the run. **Log** shows solver messages, and it is the first place to look if something goes wrong.



Figure 5. The Infos tab during the run on an RTX 5080.

Find your results

The **Output path** in **Infos** shows where this run writes its files. By default this is a `results` folder beside the setup file.

Four icons at the top left of the **Simulate** stage handle results:

Icon	What it does
Write results	Pauses and writes the current state as VTK XML, which you can open in ParaView
Screenshot	Saves the viewport as a PNG image
Copy image	Copies the viewport image to the clipboard
Open results folder	Opens the folder in your file manager

The browser version cannot write results to disk.

Change something and run again

CFD work is a loop of changing a setting and comparing runs. Try it with the grid:

1. Click **Prepare**, set **Grid cell size** to `0.01` and press `Enter`. The predicted memory rises from 0.93 GiB to 1.14 GiB. A `*` after the file name in the title bar marks unsaved changes.
2. Click **Simulate**. **Restart** has turned amber, which means the simulation on screen still uses the old setting.
3. Click **Restart**, then **Run** if the simulation is paused.

The new run has 240,018 particles, eight times as many as before, because halving the cell size doubles the cells in each direction. The time step also gets smaller. The 10 s run now takes about three minutes on the same GPU, against a little over one before.

Compare the two runs before you trust either. If the result you care about changes with the grid, the coarser grid was not fine enough.

An amber note may appear about secondary phase properties. It tells you that this variant keeps the air properties in the setup but does not simulate the air.

To keep your change, go back to **Scene** or **Prepare** and click **Save** or press `Ctrl+S`.

Keep Prelimina up to date

Click the gear icon at the top left and choose **Update**. A coloured **Update available** entry appears in the same menu when a new version is out. The same menu has **Contact support**, which starts an email containing your system details, and **About**, which shows the version, changelog and licences.

What to try next

- Open the other examples. `drop_in_tank` is a small quick case, and `dam_break_marin` is a 3.22 m dam break with an obstacle. `sloshing_kishev` reproduces a published experiment at a fine grid, so expect a much longer run.
- See how similar cases compare with measurements in Applications.
- See what each feature can do today in Capabilities.

The workspace

This chapter explains what is on screen and how the parts work together. To follow along, open the `sloshing_tank` example from Getting started.

The main window

The window has the same layout in every stage: tools along the top, settings on the left, the 3D view in the middle and a time bar along the bottom.

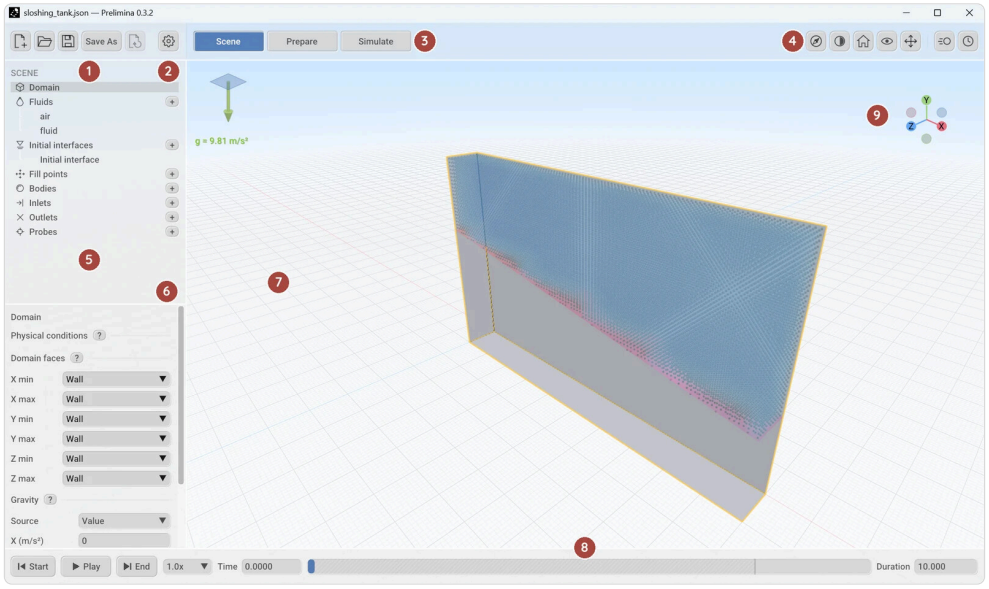


Figure 6. The main window in the Scene stage, with the domain selected.

	Area	What it does
1	File tools	New, Open, Save, Save As and Restore. In Simulate these are replaced by the result tools.
2	Settings menu	Application settings, updates, support and version details.
3	Stage tabs	Switch between Scene, Prepare and Simulate.
4	View tools	Interface theme, camera framing, view options and extra panels.
5	Scene tree	Everything in the setup, grouped by type.
6	Properties	Settings for whatever is selected in the tree.
7	Viewport	The 3D view of the setup or the results.
8	Time bar	Time, duration and playback controls.
9	View gizmo	Shows the orientation. Click an axis to look along it.

The green arrow at the top left of the viewport shows the direction and size of gravity.

The three stages

Each stage changes the left panel and the tools around it.

Stage	Left panel	Top-left tools	Time bar
Scene	Scene tree and Properties	File tools	Scrubs and plays the preview
Prepare	Solver and numerical settings	File tools	Scrubs and plays the preview
Simulate	Display settings	Result tools	Runs the simulation

You can change stage at any time, and doing so never starts, stops or resets a simulation. A run keeps going while you work in **Scene** or **Prepare**, and a pulsing green dot on the **Simulate** tab shows that it is running. A grey dot shows that it has finished.

Changes in **Scene** and **Prepare** do not affect a simulation that has already started. They edit the setup, and an amber marker on the **Simulate** tab and on **Restart** tells you the running simulation uses an earlier version. Click **Restart** to apply your changes. The simulation then starts again from t = 0.

Setup files

A setup file is a JSON file that holds everything in **Scene** and **Prepare**. It does not hold results. These go in a `results` folder beside the setup file.

Tool	Shortcut	What it does
New	`Ctrl+N`	Starts an empty setup
Open	`Ctrl+O`	Opens a setup file
Save	`Ctrl+S`	Saves to the current file
Save As	`Ctrl+Shift+S`	Saves to a new file. Available once the setup has a file.
Restore		Goes back to the last applied setup

A dot on the **Save** icon and a `*` in the title bar mean there are unsaved changes. Prelimina asks whether to save them before it replaces the open setup. The **Save** tools appear only in **Scene** and **Prepare**.

Keep meshes and motion files with your setup file. Paths inside a setup are relative to that file, so the folder can be moved as a whole.

> **There is no undo.** Save before a large change. **Restore** returns to the setup used for the latest simulation start or restart, or to the setup from New or Open if nothing has run yet. It asks first, and it leaves the saved file unchanged. Saving does not move this restore point. In **Prepare**, a changed setting also has a small revert button beside it.

Select and edit

The Scene tree has one group for each kind of item: **Domain**, **Fluids**, **Initial interfaces**, **Fill points**, **Bodies**, **Inlets**, **Outlets** and **Probes**. Click **+** beside a group to add an item.

Click a row to select it, or click an object in the viewport while in **Scene**. **Properties** then shows its settings, and the object is outlined in the viewport. Most sections of **Properties** have a **?** button with a short explanation.

To move a selected object in the viewport, choose a manipulation mode from the toolbar or with a key:

Key	Mode
`Q`	Select
`W`	Move
`E`	Rotate
`R`	Scale

Fill points and probes can only be moved. Keys do nothing while you are typing in a field.

View tools

The tools at the top right depend on the stage.

Tool	Stages	What it does
Explore setup (compass)	Scene, Prepare	Starts the setup tour
Interface theme (half disc)	All	Switches between the light and dark interface
Frame (house)	All	Frames the selection, or everything if nothing is selected
View options (eye)	All	Opens the list below
Manipulation mode	Scene	Selects Select, Move, Rotate or Scale
Motion Inspector	Scene	Shows or hides the Motion Inspector panel
Timeline Editor	Scene	Shows or hides the Timeline Editor
Live diagnostics (graph)	Simulate	Shows or hides the Log, Infos and graph tabs

A coloured dot on the diagnostics tool means the log has an unread warning or error.

View options controls what the viewport draws. None of these settings changes the physics.

Option	What it does
Filled volume	In Scene and Prepare, previews where the fluid starts: blue for liquid, amber for gas, red for conflicting assignments. This is a coarse preview.
Objects	Draws objects hidden, as edges, transparent or opaque
Particles, Free surface	In Simulate, shows or hides these layers
Sky, Grid	Sets the background. The grid can be hidden, Cartesian or polar.
Cut away all objects	Hides the near faces of obstacles as well as containers
View gizmo	Shows or hides the gizmo
Rotate around picked point	Orbits around the surface under the pointer
Rotate horizon with domain	Turns the horizon with a moving domain
Orthographic	Uses a parallel view. The `0` key does the same.

Boundaries have their own colours: grey for walls, violet for symmetry, green for inlets and red for outlets.

Simulation preview

While you work in **Scene** and **Prepare**, Prelimina runs a coarse FLIP simulation of your setup in the background. Grey hatching on the time bar shows how far it has got. Use **Play** or drag the time bar to watch it.

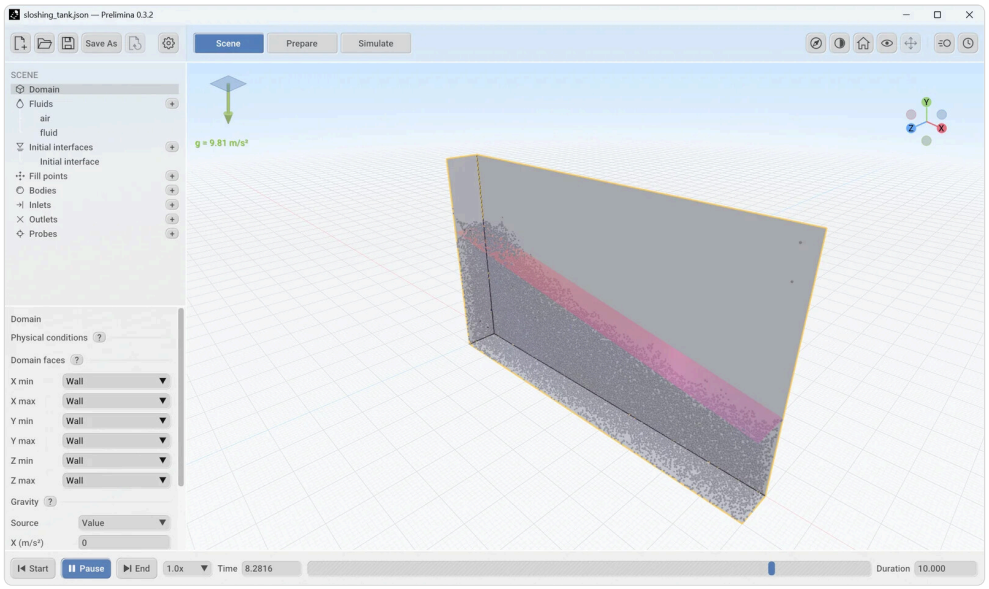


Figure 7. The preview at t = 8.3 s in Scene. It updates as you change the setup.

The preview shows quickly whether a setup behaves as you intended, for example whether the liquid starts in the right place or a body moves the right way. It restarts when you change something it depends on. Nothing from it is saved, and **Simulate** does not use it.

> **Don't take numbers from the preview.** It always uses FLIP, even if your setup uses another solver, and it runs with its own coarse settings. Thin sheets and splash volume in particular are only indicative. Use **Simulate** for results.

A simulation and the preview cannot share the GPU. Once a simulation has started, the preview pauses and the time bar offers **Release run…**. That stops the simulation and frees its memory so the preview can continue. Any results you have not exported are discarded, and the next **Run** starts from t = 0.

To turn the preview off, clear **Compute simulation preview** in the settings menu. This is worth doing on a GPU with little memory.

Explore a setup

The tour shows what a setup changes from the solver defaults, which is the quickest way to understand a file from someone else. It takes you through these settings in steps, such as domain and boundaries, then fluids, then solver method. Everything else is dimmed.

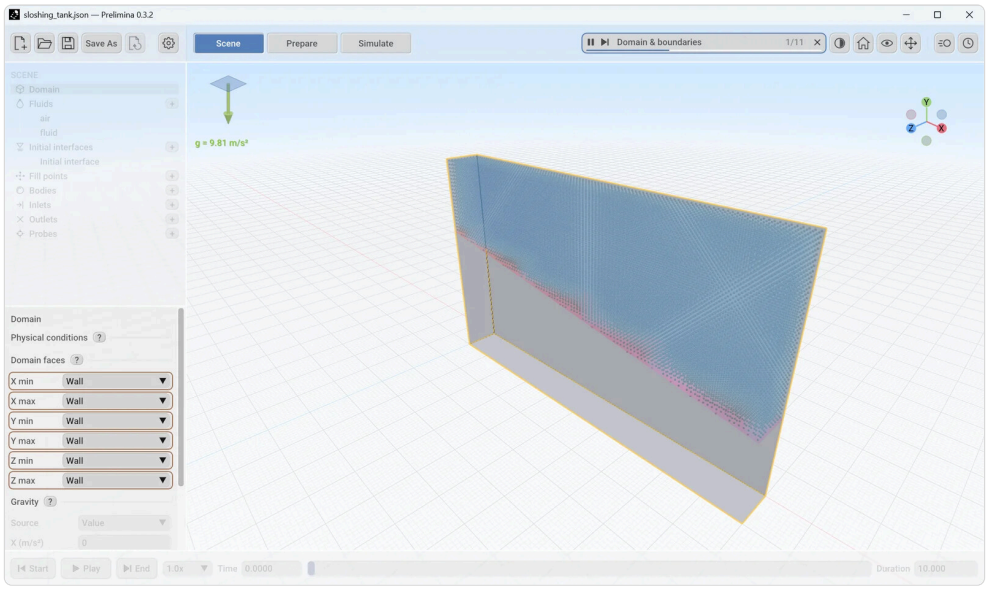


Figure 8. The tour on the first of 11 steps. The domain faces are highlighted and the rest of the window is dimmed.

Start the tour in one of three ways:

- Click **Explore setup** in the banner that appears when you open a file.
- Click the compass icon in the view tools.
- Choose **Explore setup** in the settings menu.

The tour moves on by itself. Its controls sit in the toolbar: pause or play, next step, the step counter, and a close button. `Esc` also ends the tour. Hover over the step title to see a table of the changed settings, and over a highlighted row to see its default.

You can keep editing during the tour, and editing pauses it. The tour is available in **Scene** and **Prepare**, and going to **Simulate** ends it.

Settings menu

Click the gear icon to open this menu.

Item	What it does
Compute backend	Shows the compute backend. Released builds use Vulkan.
Explore setup	Starts the setup tour, in Scene and Prepare
Dark interface	Switches the interface theme
Compute simulation preview	Turns the preview on or off
Unlock beta features	Enables features that are not ready for general use, including the FVM and FIRM solvers and the two-phase variants. You don't need it for this guide.
Update	Checks for a new version
Contact support	Starts an email containing your system details
About	Shows the version, system details, changelog and licences

Keyboard shortcuts

Keys	Action
`Ctrl+N`	New setup
`Ctrl+O`	Open a setup
`Ctrl+S`	Save
`Ctrl+Shift+S`	Save As
`Space`	Run or pause in Simulate, play or pause the preview in Scene and Prepare
`Home`	Frame the selection, or everything if nothing is selected
`Shift+Home`	Frame everything
`O`	Switch between perspective and parallel view
`Q`, `W`, `E`, `R`	Select, Move, Rotate, Scale, in Scene
`Esc`	Close the tour or cancel the current placement or transform

When the time bar in **Scene** or **Prepare** has focus, the arrow keys step through time, and `Home` and `End` jump to the start and end.

The scene

The **Scene** stage holds the physical problem. This chapter starts with a complete example that you build yourself, and then explains each part of the Scene tree.

Item	What it holds	Status
Domain	Face conditions, gravity and motion of the whole frame	Implemented
Fluids	Density, viscosity, surface tension and contact angle	Density and viscosity implemented, the rest experimental
Initial interfaces, Fill points	Where the liquid starts	Implemented
Bodies	Tanks, obstacles and imported geometry	Implemented
Inlets, Outlets	Flow through the boundary	Experimental on domain faces, in development on surfaces
Probes	Measurements recorded during the run	Implemented

Status means the same as on the Capabilities page. Experimental features work, but their interface and defaults may still change.

Prelimina uses SI units throughout: metres, seconds, kilograms and pascals. A new setup has gravity along −Y, so Y is up. You can point gravity in any direction, but this guide keeps the default.

Build a sloshing tank

You will build a tank 1 m long, 0.6 m high and 0.3 m wide, fill it to 0.2 m, shake it from side to side and measure the pressure near one end. This takes about 10 minutes.

Make the tank

1. Click **New** or press `Ctrl+N`.
2. Click **+** beside **Bodies** and choose **Box**. A new box is a 1 m cube centred on the origin.
3. In **Properties**, set **Body role** to **Container**, so that the fluid stays inside the box. Set **Name** to `Tank`.
4. Set **Position** Y to `0.3`. Press `Enter` after each value.
5. Set **Scale** to `1`, `0.6` and `0.3`.

The tank now runs from y = 0 to y = 0.6 m, with its floor on the ground grid. The near walls are cut away so that you can see inside.

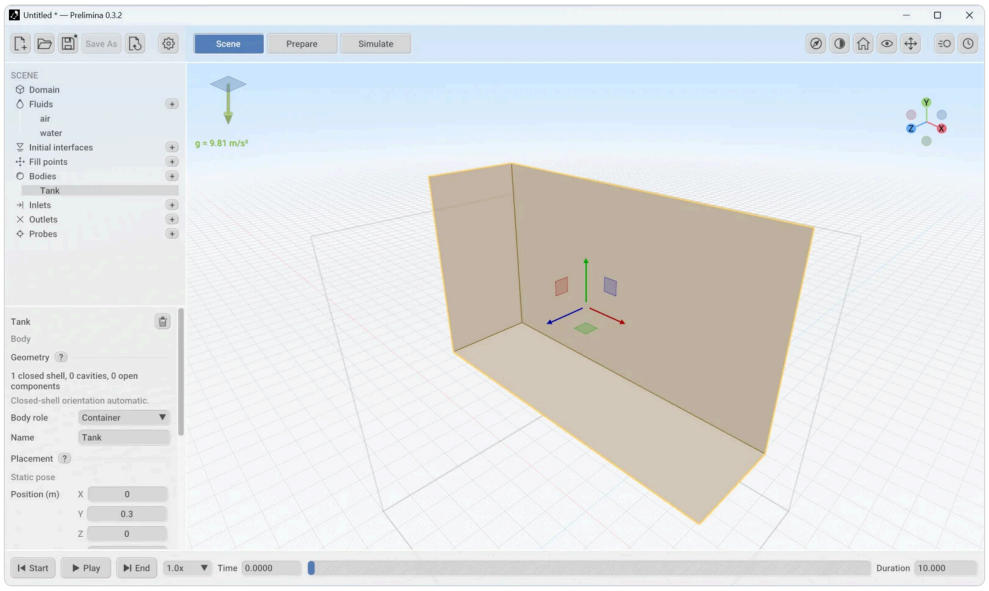


Figure 9. The new tank, with its role set to Container.

Add the liquid

1. Click **+** beside **Initial interfaces** and choose **plane**. A plane is the flat surface between liquid and empty space.
2. Set **Position** to `0`, `0.2` and `0`.

The plane starts at (0.5, 0.5, 0.5), which may be outside your geometry. That is why you move it first. If an error about `initial_candidate_unresolved` appears, the plane is not yet inside the tank.

Fill volume stays on **Automatic**, which fills the side of the plane that gravity points to. The liquid appears in blue, and the simulation preview starts in the background. The plane is drawn at a fixed size, but it separates the liquid across the whole tank.

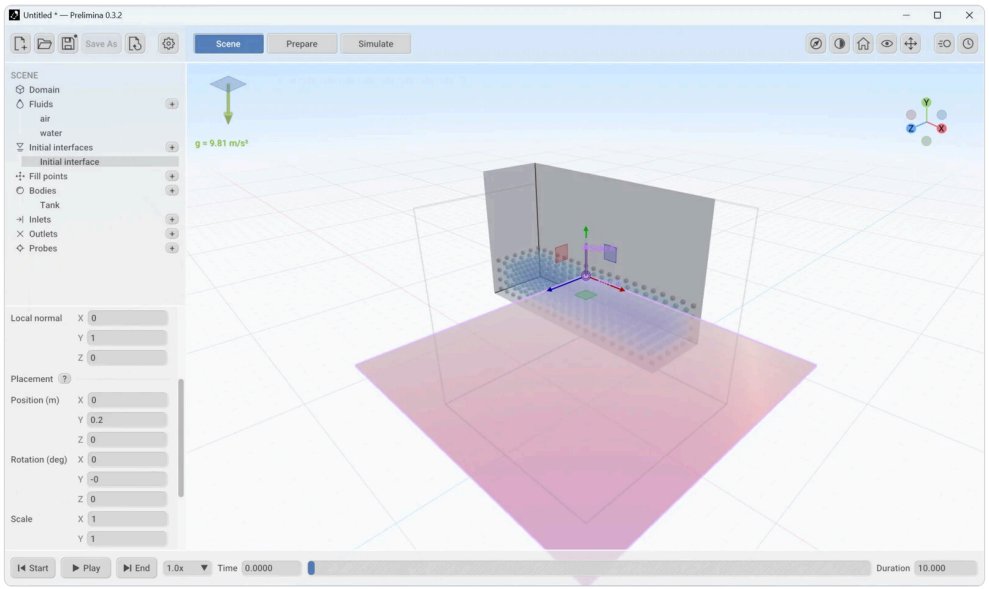


Figure 10. The plane at y = 0.2 m. The blue volume and the preview particles below it show where the liquid starts.

Make it move

1. Select **Tank** in the Scene tree.
2. Scroll down in **Properties** and click **Add or edit motion…**. The Timeline Editor opens.
3. Expand **Harmonic Translation**. Its direction is already X.
4. Set **Amplitude** to `0.01` and **Frequency** to `0.66`.

The tank now moves ±10 mm along X. The panel shows the peak speed, 0.041 m/s, and the tree marks the tank as `(animated)`.

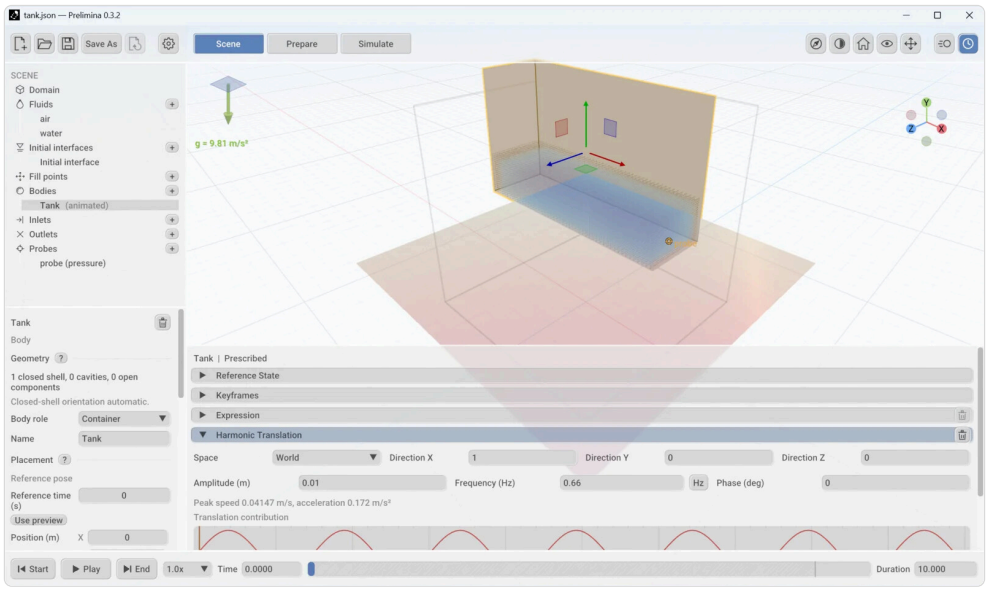


Figure 11. Harmonic motion on the tank, with the signal drawn below the settings.

The value 0.66 Hz is close to the lowest natural sloshing frequency of this tank, so the waves should grow. For a tank of length L filled to depth h, that frequency is √(g·k·tanh(k·h)) / 2π, with k = π / L.

Add a probe

1. Click **+** beside **Probes** and choose **Pressure probe**.
2. Set **Position** to `0.46`, `0.05` and `0`.
3. Click **Apply measurement**.

The probe samples a sphere of 0.02 m radius, close to the end wall and 0.05 m above the floor.

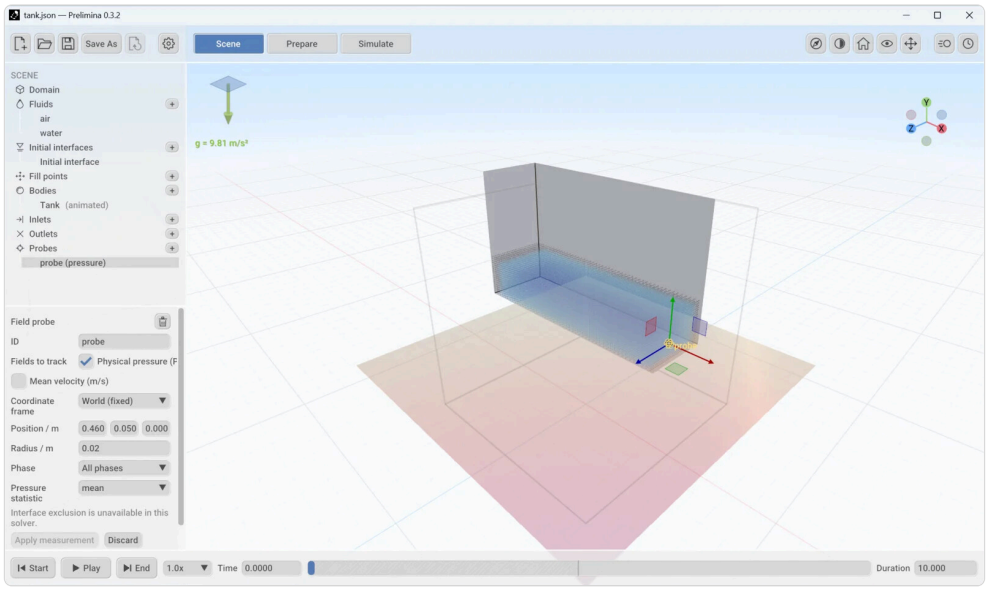


Figure 12. The probe selected, near the floor at the right-hand end of the tank.

Save and run

1. Press `Ctrl+S` and save the setup in a new folder of your own. The results will go in a `results` folder beside it.
2. Click **Prepare** and set **Grid cell size** to `0.02`. New setups start at 0.1 m, which is too coarse for this tank.
3. Click **Simulate**, then **Run**.
4. Click the graph icon to open the diagnostics, then click **Flow**. Choose **Pressure / probe** to follow the measurement.

The run has 60,000 particles. It took under half a minute on an NVIDIA RTX 5080, at about 65 steps per second.

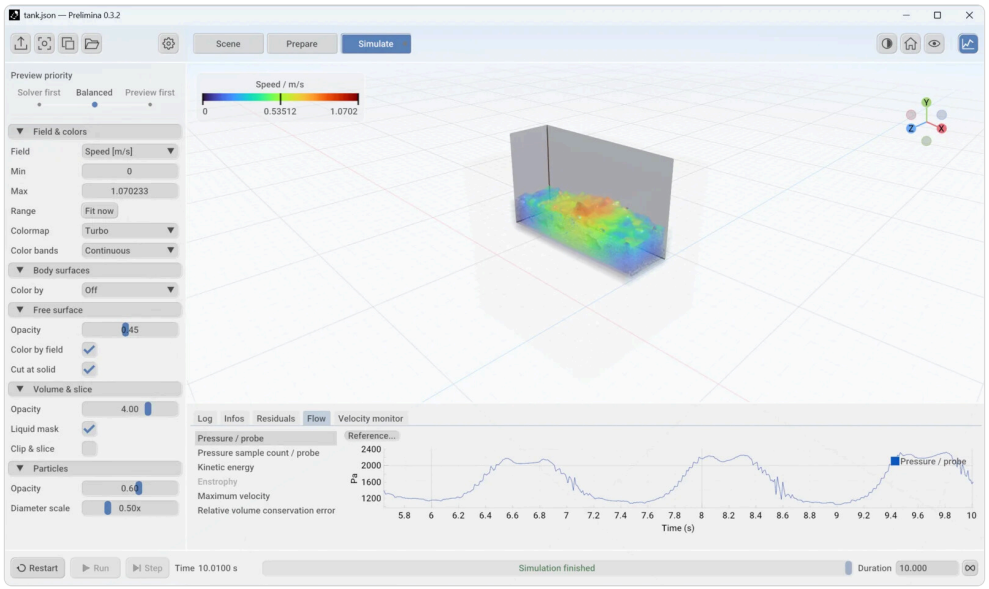


Figure 13. The finished run. The pressure graph shows the sloshing period, and the colours show the speed.

The probe starts near the hydrostatic pressure of 1,470 Pa for 0.15 m of water. It then swings with a period of about 1.5 s, and between 8 and 10 s it ranges from about 1,000 to 2,400 Pa.

The waves are still growing when the run ends, so run longer if you need the steady amplitude.

Prelimina writes the probe data to `results/probes/measurements.csv` without being asked. Each new run starts a new numbered file, so earlier runs are kept.

> **This is a first run, not a result.** A new setup uses a fixed time step of 0.01 s, which is too long for this tank. Even a still tank goes unstable after about 5 s at this step. Prepare a run corrects it and shows how to check a setup.

The domain

The domain is the region the solver works in. How you define it depends on your setup.

Without a fixed size. A new setup has none. The solver sizes its grid from the liquid and the geometry, which is what the tank above uses. The thin grey box in the viewport is a 1 m placeholder and limits nothing. Leave the face settings alone in this kind of setup, because they would attach to that placeholder.

With a fixed size. You need this for domain walls, inlets, outlets and periodic faces. Prelimina 0.3.2 has no field for the domain size, so you add it to the setup file with a text editor. Save and close the setup first, then add `lower` and `upper` to the existing `domain` block:

```
"domain": {
  "lower": [0, 0, 0],
  "upper": [1, 0.6, 0.3],
  ...
}
```

Open the file again. The domain box now has that size, and the six faces in **Properties** apply to it.

Face conditions

Select **Domain** to set the condition on each face.

Condition	What it does
Wall	Closed face. In FLIP the liquid slips along it freely.
Symmetry	Mirror plane. FLIP treats it approximately, as a wall.
Periodic	Connects this face to the opposite one
Inlet	Flow in through the whole face. See Inlets and outlets.
Open (pressure outlet)	Open face at a set pressure. See Inlets and outlets.
Solver default	No condition saved. The solver's default applies.

A wall face also has **Edit prescribed velocity in Timeline…**. FLIP saves this velocity but does not apply it, and a warning says so. To move a wall, use a body or reference-frame motion.

Gravity and duration

Gravity has three components in m/s². Each is a number or an expression in time `t`, or you can choose **Keyframed** in **Source** to draw a history. **Duration** in the time bar sets how long the simulation runs.

Reference-frame motion

Prescribed motion under **Domain** moves the whole frame, including the grid, and the solver adds the matching inertial forces. It uses the same Timeline Editor as a body. The `sloshing_kishev` example uses it, with the tank's acceleration read from a CSV file.

With body motion the tank moves through a fixed grid. With reference-frame motion the tank stays still relative to the grid, which keeps the walls on the same cells throughout the run.

Fluids

A new setup has two fluids, **water** and **air**. Click one to edit it, or click **+** to add a Water or Air preset.

Setting	Meaning
Name	Letters, digits, dots, underscores and hyphens. References update when you rename.
Density	kg/m³, must be positive
Dynamic viscosity	Pa·s. Zero means inviscid.
Rheology	Newtonian, or a non-Newtonian model. Non-Newtonian models are in development.

Click **Fluids** itself for the settings shared between them. **Primary phase** selects the liquid, and **Automatic** chooses the denser fluid. **Surface tension** and **Contact angle** are experimental.

> **Viscosity needs a solver setting.** FLIP ignores the viscosity here unless viscosity is enabled in **Prepare**, and the Log warns you about this. The default **Single phase + void** variant also leaves the air out of the simulation, although its properties stay in the setup.

Bodies

Click **+** beside **Bodies** to add a **Box**, **Sphere** or **Cylinder**, or choose **Mesh file…** to import geometry.

Every body has a role:

Role	Fluid is	Use it for
Obstacle	Outside the surface	Baffles, hulls, gates and other objects in the flow
Container	Inside the surface	Tanks and vessels

Placement sets the position, the rotation in degrees and the scale. You can also use the Move, Rotate and Scale tools described in The workspace. **Scale** works around the body's position, so a box stays centred where you put it.

Import a mesh

Prelimina reads STL, OBJ, PLY, glTF and GLB files, and you do not need a volume mesh.

- Coordinates are read as metres. A mesh in millimetres needs a **Scale** of 0.001.
- The body keeps the coordinates in the file, so **Position** starts at 0.
- The view zooms in to the new body. Press `Shift+Home` to see everything again.
- Keep the file with your setup. **Advanced** has **Embed compressed fallback**, which stores a copy in the setup file.

Geometry in **Properties** reports what Prelimina found, for example `1 closed shell, 0 cavities, 0 open components`. A container should be closed. Prelimina corrects the direction of the faces, but it does not close cracks or weld nearby points, and it rejects non-manifold geometry.

Bodies do not move together

Each body has its own motion, and one body cannot follow another. A baffle inside a moving tank needs the same motion as the tank. With reference-frame motion this problem does not arise, because everything moves with the frame.

Fluid-driven bodies, which the flow moves, are experimental. They have no controls in **Scene** yet, and this guide does not cover them.

Initial liquid

Two kinds of item define where the liquid starts. **Initial interfaces** are surfaces that divide the space, and **Fill points** choose which of the divided regions hold liquid. Neither is a wall: an interface has no effect once the simulation starts.

Initial interfaces

Click **+** beside **Initial interfaces** to add a **plane**, **sphere**, **box** or **expression**, or use **Import mesh…**. Rotate a plane to start with a tilted surface.

Fill volume chooses which side of the interface fills:

Fill volume	Fills
Automatic	The inside of a closed shape, otherwise the side that gravity points to
Along gravity	The side that gravity points to
Side A, Side B	The side the arrow points to, or the other side
Custom direction	The side a direction you enter points to
None	Nothing. The interface only divides the space.

With no interfaces and no fill points there is no initial liquid.

Fill points

Use fill points when automatic filling cannot work out what you mean, for example with several compartments or nested shapes.

To add one, select an interface and click **Add fill point…** at the bottom of **Properties**. Under **Suggested position**, choose a side. Under **Pick on surface**, choose a side and click the surface in the viewport. You can also click **+** beside **Fill points**.

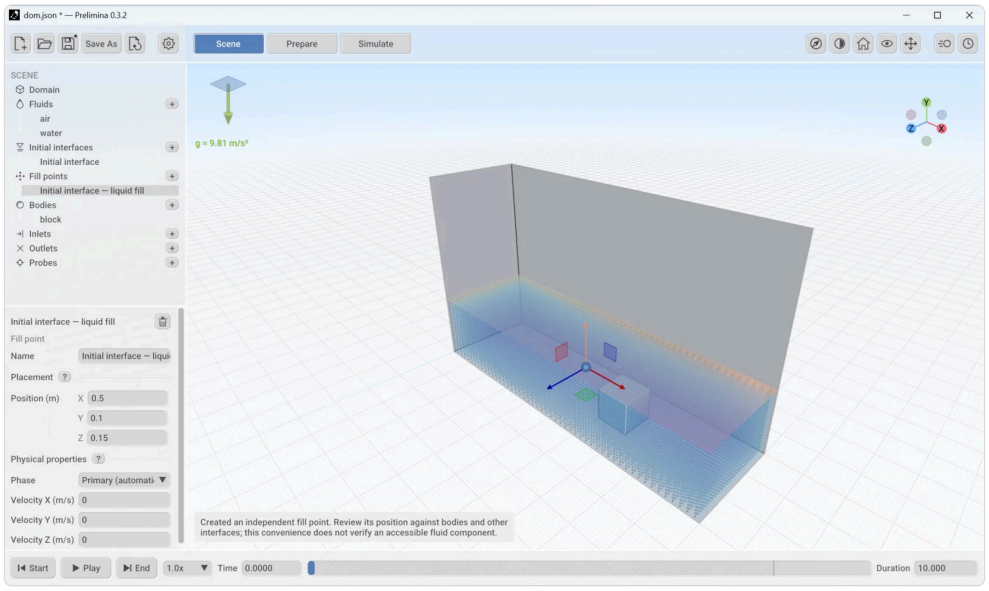


Figure 14. A fill point below the waterline, in a domain with an imported block. Its name comes from the interface.

A fill point marks a connected region of space, which then fills completely up to the nearest walls and interfaces.

> **One fill point replaces all automatic filling.** As soon as the scene has a fill point, only regions with fill points fill, so every compartment needs its own. Deleting the last fill point brings automatic filling back. Suggested positions are not checked against your geometry, so look at where the point lands.

Check the fill

Filled volume in View options shows the result in **Scene** and **Prepare**: blue for liquid, amber for gas, and red where assignments conflict. It is a coarse preview, so it can differ from the real fill near curved walls and in narrow gaps. The preview particles are a useful second check.

Motion

Select a body or **Domain**, then open the Timeline Editor with **Add or edit motion…**, **Edit timeline**, or the clock icon in the view tools. The Motion Inspector, next to the clock icon, lists the position, velocity and acceleration at the current time.

Reference State holds the starting pose. The motion sources add to it:

Source	Use it for
Keyframes	Hand-drawn histories. Move the time bar, then click **Add Key**. Translation can use position, velocity or acceleration keys.
Expression	A formula in time `t` for position, rotation, velocity or acceleration
Harmonic Translation	Sinusoidal movement, with direction, amplitude, frequency and phase
Harmonic Rotation	Sinusoidal rotation about an axis
From file	A CSV file with time in the first column

Space selects world axes or local axes, which rotate with the body. From a file, you can read a full transform, or one of position, rotation, velocity or acceleration. For a full transform, **CSV format** is `t,pos,quat` or `t,pos,euler`.

Click **Play** in **Scene** or **Prepare**, or drag the time bar, to check the motion before you run.

Inlets and outlets

> **Experimental.** Face inlets and outlets need a fixed domain size. Mesh-surface inlets (**Rectangle**, **Circle** and **Mesh file…**) are still in development. Start with short runs and read the Log.

To set up a face, select **Domain** and choose **Inlet** or **Open (pressure outlet)** in the list for that face. It then appears under **Inlets** or **Outlets**. You can also use **+** beside those groups.

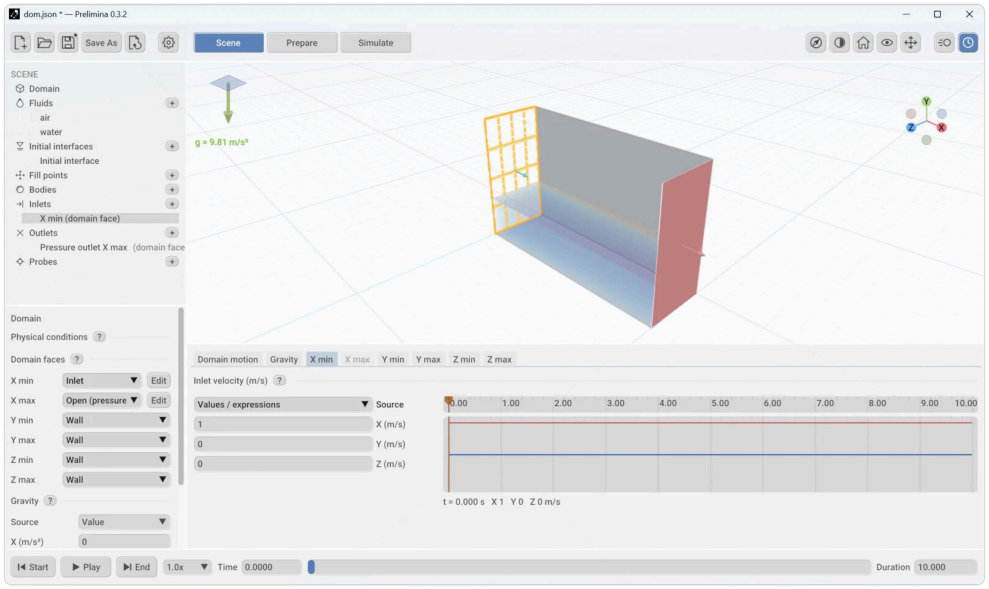


Figure 15. An inlet on X min and an outlet on X max, with the inlet velocity in the Timeline Editor.

A new inlet flows inward at 1 m/s. Change this in the Timeline Editor, as a number, an expression or keyframes. Inlets need a shorter time step than a new setup uses: at the default 0.01 s, inlet runs stopped with an error within 5 s. Use the settings for inlets and outlets. Adding an inlet can bring back the **Optimizing the simulation for your device** message for that run.

An outlet has these settings:

Setting	Meaning
Pressure	The pressure on the face, in Pa
Pressure type	Static, or Total to include the dynamic pressure
Backflow	Clamp reverse flow, or Allow exterior inflow
Exterior velocity	Velocity of the flow that comes back in when backflow is allowed

The pressure is the same over the whole face, so an outlet below a free surface drains the liquid above it.

Probes and measurements

Click **+** beside **Probes** to add a measurement. Everything here is recorded to CSV automatically, whether or not the diagnostics are open.

Item	What it records
Pressure probe	Pressure in a small region of fluid
Velocity probe	Mean velocity in a small region of fluid
Tracer	The path of a point carried by the flow
Body measurements	The motion of a body, and optionally the fluid force and moment on it

A probe can track pressure, velocity or both. **Coordinate frame** fixes it in the world or attaches it to a body so that it follows the motion. **Phase** limits it to liquid or gas, and **Pressure statistic** selects the mean, minimum or maximum.

Set the position in three ways: type it, drag it with the Move tool, or click **Pick position in viewport**. Click **Apply measurement** to confirm your changes.

> **A probe is not a wall sensor.** It averages the fluid around a point. To compare with a wall pressure tap, keep the probe in the liquid close to the wall, and check how much the result changes with its position and radius.

Next steps

- Set the time step and grid properly in Prepare a run.
- See how a body, its motion and its probes are set up in a measured case in Applications.

Prepare a run

The **Prepare** stage holds the numerical choices: how finely the flow is resolved and how the solver advances in time. These settings decide whether a result means anything, and the defaults of a new setup are not always safe. This chapter starts by correcting the tank from The scene, then explains why each change matters.

All measurements here come from Prelimina 0.3.2 on that tank, with FLIP on an NVIDIA RTX 5080. They show how to test your own case. They are not general limits.

Correct the tank

Open the tank you saved in the last chapter and make these changes.

1. Click **Prepare**. Open the list that shows **Resolution** and choose **Time stepping**.
2. Change the mode from **Fixed** to **Adaptive**, type `0.4` for **Target CFL** and press `Enter`.
3. Set **Maximum adaptive time step** to `0.0025` and press `Enter`.

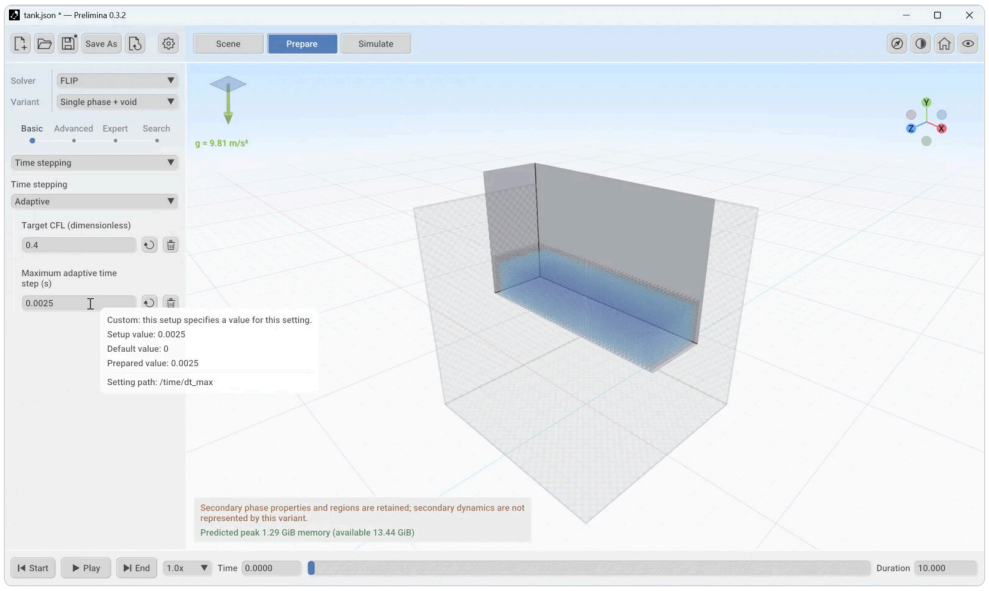


Figure 16. Adaptive time stepping. Hovering over a field shows its value, its default and its setting path.

4. Move the slider at the top of the panel from **Basic** to **Advanced**, then choose **Transport & stabilization** in the list.
5. Set **PBD iterations before pressure projection** to `2` and press `Enter`.

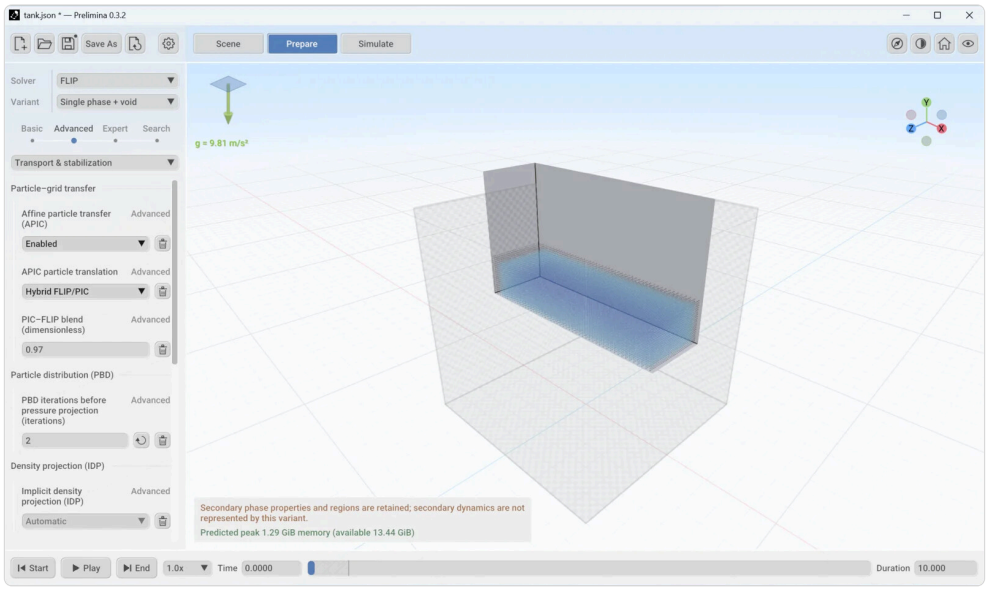


Figure 17. PBD iterations in the Advanced level. The arrow beside the field reverts this change.

6. Set **Duration** in the time bar to `20`.
7. Click **Simulate**, then **Restart** if it is available, and **Run**.

The run takes 8,000 steps instead of 1,000, and 20 simulated seconds took about 100 s. From 12 s to the end, the probe swings between about 900 and 2,400 Pa, with some short spikes above that. Those spikes need care if you are interested in peak pressure.

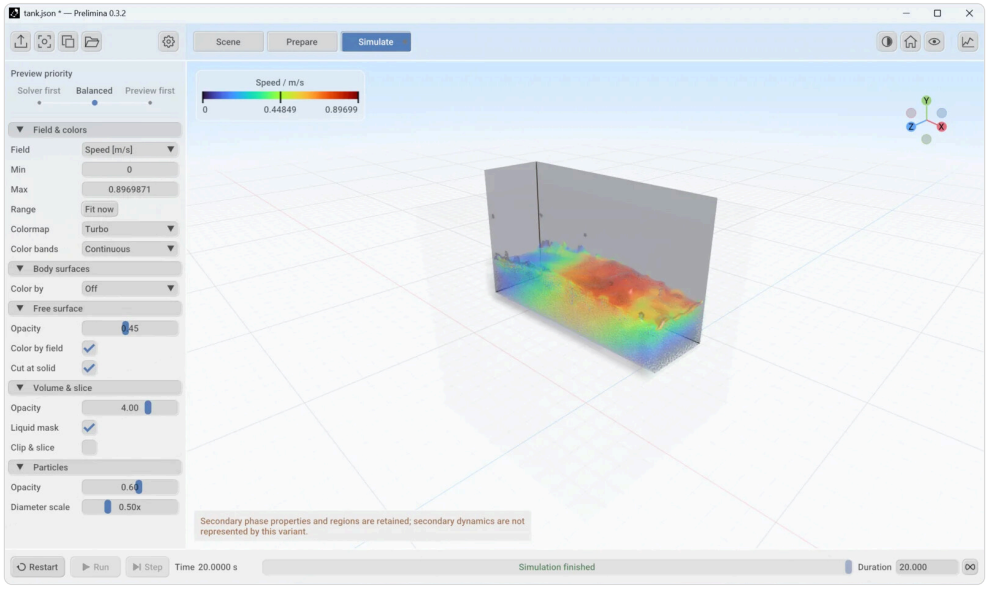


Figure 18. The corrected tank after 20 s, with the colour range fitted to the speed.

The rest of this chapter explains these two changes, then shows how to check a setup of your own.

Read the panel

The top of the panel shows **Solver** and **Variant**. This guide uses **FLIP** with **Single phase + void**, which simulates the liquid and treats the space above it as empty. Other solvers and variants are locked until you choose **Unlock beta features** in the settings menu, and this guide does not cover them.

The slider sets how much is shown, and the list below it selects a section.

Level	What it shows
Basic	Grid cell size and time stepping only
Advanced	The settings you are most likely to change, such as particle capacity and PBD
Expert	Everything in the section. Most of it should stay at the default.
Search	Finds a setting by name in every section and level

Section	What it holds
Resolution	Grid cell size and particle sampling
Time stepping	Step size and its limits
Pressure & convergence	The pressure solver and when it stops
Transport & stabilization	How velocities move between particles and grid, PBD and density projection
Interfaces & boundaries	Surface reconstruction and solid walls
Performance & diagnostics	Particle and grid capacity, extra diagnostics
Coupling	Links to external solvers, in Expert only

Dimmed text in a field means the solver chooses the value. Normal text means your setup specifies it. The bin icon removes your value and returns to the default, and the arrow icon reverts a recent change. Hover over a field to see its **Setup value**, **Default value** and **Prepared value**, which is what the solver will actually use. The **Setting path** names the same setting in the setup file.

Grid cell size

Grid cell size is the most important setting, and it applies in all three directions. Halving it gives eight times the cells and particles. A new setup starts at 0.1 m, so you will nearly always need to change it.

The grid has to fit the smallest thing you care about, such as the liquid depth, a gap, a baffle or a wave. In the tank, these were the results over the 0.2 m depth with the liquid at rest:

Cell size	Cells over the depth	Particles	Result
0.04 m	5	7,500	Level rose by 18 mm within 2.5 s, at every time step tried
0.02 m	10	60,000	Level held at the right step size
0.01 m	20	480,000	Level held at the right step size

In the moving tank, the finer grid changed the result. The filtered probe swing from 12 to 20 s was about 900 to 2,400 Pa at 0.02 m, and about 500 to 2,900 Pa at 0.01 m, which is 60 % larger. So 0.02 m is enough to learn how this tank behaves, but not to quote a peak pressure. That needs at least one more refinement, and the two results must then agree well enough for your purpose.

The finer run took about 6 minutes against 100 s. It has eight times the particles and, with the step reduced in proportion, twice the steps.

Particles

By default each grid cell starts with 2 particles along each axis, so 8 per cell. This is set under **Particle sampling** in **Resolution** at the **Advanced** level, and you rarely need to change it. Density projection needs an even sampling of 2 or 3 per axis.

Particle capacity is a separate upper limit, under **Performance & diagnostics**. A new setup allows 1,000,000. If the initial liquid needs more, the run stops at the start and names the setting:

```
FLIP initialization emits 3840000 particles, exceeding the particle capacity of
1000000 (flip.particles.max_particles)
```

To estimate the count before you run, divide the liquid volume by the cell volume and multiply by 8.

Time stepping

The time step is the setting most likely to spoil a result without any warning. The **Time stepping** list has three modes:

Mode	What it does
Default	Lets the solver decide
Fixed	Uses the same **Fixed time step** throughout. New setups use this, with 0.01 s.
Adaptive	Changes the step during the run to hold a **Target CFL**, up to a **Maximum adaptive time step**

The CFL number is speed × time step ÷ cell size. It tells you what fraction of a cell the fastest flow crosses in one step. At a target of 0.4, the step shrinks as the flow speeds up.

> **CFL does nothing in calm liquid.** If the flow is slow, the step grows until it reaches the maximum. So **Maximum adaptive time step** is the limit that protects a resting or slowly moving surface. It shows 0 until you set it, so always enter a value.

How long a step is too long

This test used the tank with no motion, so the correct answer is known: a flat surface and a steady hydrostatic pressure of 1,471.5 Pa at the probe.

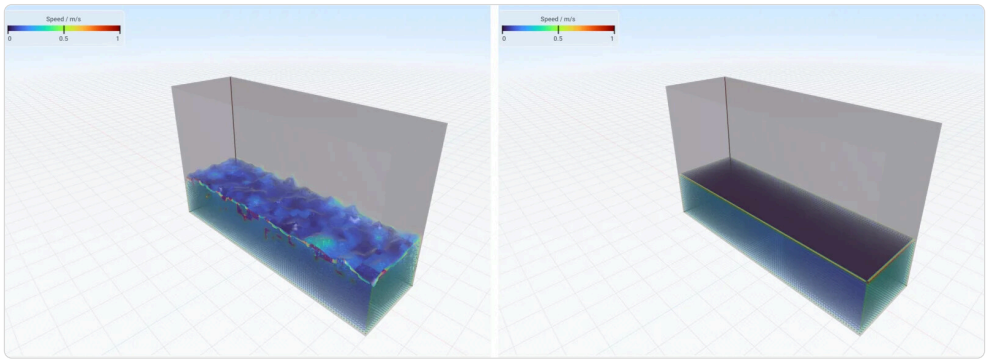


Figure 19. The same still tank at t = 10 s. Left, the default fixed step of 0.01 s. Right, a step of 0.002 s.

Cell size	Time step	Over 30 s
0.02 m	0.01 s, 0.005 s	Flat for about 5 s, then rough, with the level about 10 mm too high
0.02 m	0.0025 s, 0.002 s	Flat throughout, and the probe reads 1,471 Pa
0.01 m	0.01 s, 0.005 s, 0.0025 s	Flat for about 10 s, then rough, with the level about 5 mm too high
0.01 m	0.00125 s	Flat throughout, and the probe reads 1,471 Pa

Adaptive stepping with a target of 0.4 and a maximum of 0.01 s failed in the same way as a fixed 0.01 s, because the liquid never moves fast enough for the CFL limit to act.

The surface stayed flat when the step was no more than one eighth of the cell size, taking metres and seconds. This gives a starting point:

> **Maximum step in seconds ≈ cell size in metres ÷ 8.** That is 0.0025 s for a 0.02 m cell and 0.00125 s for a 0.01 m cell. It comes from these tests on one tank, so check it on your own case.

The instability does not show at first, so a short run can look perfectly clean. Run the check for as long as your real simulation.

A smaller step is not always better

A very small step causes its own trouble. In the moving tank at 0.02 m with no PBD, the liquid at 0.0025 s began to sink after about 15 s. At 0.001 s it had collapsed to half its depth by 20 s, in both of two runs. The step of 0.002 s came through, so this behaviour is erratic and you should not rely on a lucky value. In every collapsed run the simulation still reported success.

PBD iterations prevents this. PBD spreads out particles that have become too crowded, before the pressure step. With 2 iterations, as in the tutorial, the moving tank stayed stable at every step tried, on both grids. These are the 0.02 m results:

Moving tank, 0.02 m, 20 s	PBD 0 (new setups)	PBD 2
Step 0.005 s	Stable	Stable
Step 0.0025 s	Sinking after 15 s	Stable
Step 0.001 s	Collapsed	Stable

PBD has a price. In the still tank with 2 iterations, the level stayed within 2 mm at a step of 0.0025 s, but the probe read 1,566 Pa at 0.02 m, which is 6 % high. At 0.01 m it read 4 % high at 0.0025 s, and correctly at 0.00125 s. So allow for an error of this size in mean pressure.

Implicit density projection (IDP), just below it, keeps the liquid at its correct density. It is on **Automatic** by default, and this is the right choice. With it off, at the default step, the level in the still tank fell by 8 % within 10 s. The Prelimina documentation recommends using it with 2 PBD iterations.

Recommended starting point

Setting	Closed tank, moving liquid	Inlets and outlets
Time stepping	Adaptive	Adaptive
Target CFL	0.4	0.4
Maximum adaptive time step	Cell size ÷ 8	Cell size ÷ 8
PBD iterations	2	0
Implicit density projection	Automatic	Automatic

Inlets are the exception on PBD. With these settings and no PBD, a 1 × 0.6 × 0.3 m channel ran 10 s cleanly at inflows of both 1 m/s and 0.2 m/s, with speeds of 3 to 4 m/s, about what water reaches falling through the channel height. With PBD at 2, speeds jumped to between 8 and 18 m/s. With the default fixed step of 0.01 s, both inflows stopped with this error in the Log, after speeds of 28 m/s:

```
FLIP density projection candidate validation failed before commit
```

Turning off density projection stops that error, but the speeds stay at 28 m/s, so the run is still wrong. Reduce the step instead.

Pressure and transport

Leave these at their defaults unless you have a specific reason.

Pressure & convergence controls how accurately each step enforces incompressibility. The solver is **PCG**, with a limit of 60 iterations. To see how it is doing, open the **Residuals** tab during a run. **Pressure solve residual** should sit below its **Target** line. It stayed below it in the corrected tank.

Transport & stabilization has **PIC–FLIP blend**, 0.97 by default. A higher value keeps more detail in the flow and a lower value adds numerical damping. Do not use it to tune a result to a measurement.

Check the memory

The line below the viewport predicts the peak GPU memory for the run:

```
Predicted peak 1.29 GiB memory (available 13.44 GiB)
```

Available leaves a margin for the driver and other programs, so it is less than the memory on the card. Here it is 13.44 GiB on a 16 GB card. The line is green when there is room, amber when the margin is small or the memory cannot be read, and red when the run will not fit.

> **The prediction follows the capacity, not your liquid.** It is calculated from **Particle capacity**, plus the grid if the domain has a fixed size. The tank has no fixed domain size, so the line stayed at 1.29 GiB from a 0.02 m cell to a 0.005 m cell, although the finer grid needs 64 times the particles and cannot start at the default capacity. A green line is a check on the capacity you set, not proof that the run will work. Keep **Particle capacity** close to what you need.

Check a setup before a long run

A run that finishes is not a run that is right. Prelimina reported success for every collapsed tank above. Before you trust a result, work through these checks.

1. **Rest test.** Save a copy of the setup and stop the motion, for example by setting **Amplitude** to `0`. Run it for the full duration. The surface should stay flat, and a probe should read density × gravity × depth.
2. **Compare the level.** Look at the liquid at the start and end of the real run. If it has clearly sunk or swollen, stop and change the settings.
3. **Check the speeds.** In the **Flow** tab, look at **Maximum velocity**. Estimate a realistic value first. Water falling through a height H reaches about √(2·g·H), which is 3.4 m/s for 0.6 m. Far higher speeds that persist mean a numerical problem, not physics.
4. **Halve the time step.** The result you care about should not change much. Keep PBD on for this, or a small step may fail as described above.
5. **Halve the cell size and the step together.** If the result moves more than you can accept, the coarser grid was not fine enough.

> **The volume graph cannot see this. Relative volume conservation error** in the **Flow** tab counts particles and multiplies by their fixed volume. It stayed at zero while the liquid collapsed to half its depth. Use the rest test and the level check instead.

For work you will rely on, compare with a measured or analytical case first. Applications shows how this looks.

Next steps

- Read and save the results in Run and read the results.
- See which features are ready for use in Capabilities.

Run and read the results

The **Simulate** stage runs the solver and shows the flow as it develops. This chapter uses the corrected tank from Prepare a run, paused at about 10.6 s.

Decide what you need to keep before you start a long run, because Prelimina saves much less than it shows.

What	Where it goes	Kept after the run
The picture in the viewport	Screen only	No, unless you save a screenshot
Graphs in the diagnostics	Screen only, about the last second	No
Probes	`results/probes`	Yes, written during the run for all of it
Flow fields at one moment	**Write results**	Yes, when you click it
Flow fields over time	Command line only	Yes

> **There is no replay.** The time bar in **Simulate** shows progress, and you cannot drag it back. The flow on screen is the current state of the solver. To look at an earlier moment, restart and pause there, or save fields during the run.

Control the run

Control	What it does
Run, Pause	Starts or pauses the solver. `Space` does the same.
Step	Advances one time step while paused
Restart	Starts again from t = 0, using your latest changes
Duration	Sets the end time
∞	Runs with no end time until you pause. Click again to go back to the set duration.

Preview priority, at the top of the panel, shares the GPU between the solver and the picture. **Solver first** favours speed, **Preview first** favours a smooth picture and **Balanced** sits between them. It does not change the results.

Choose what to show

The panel on the left has one section for each layer of the picture.

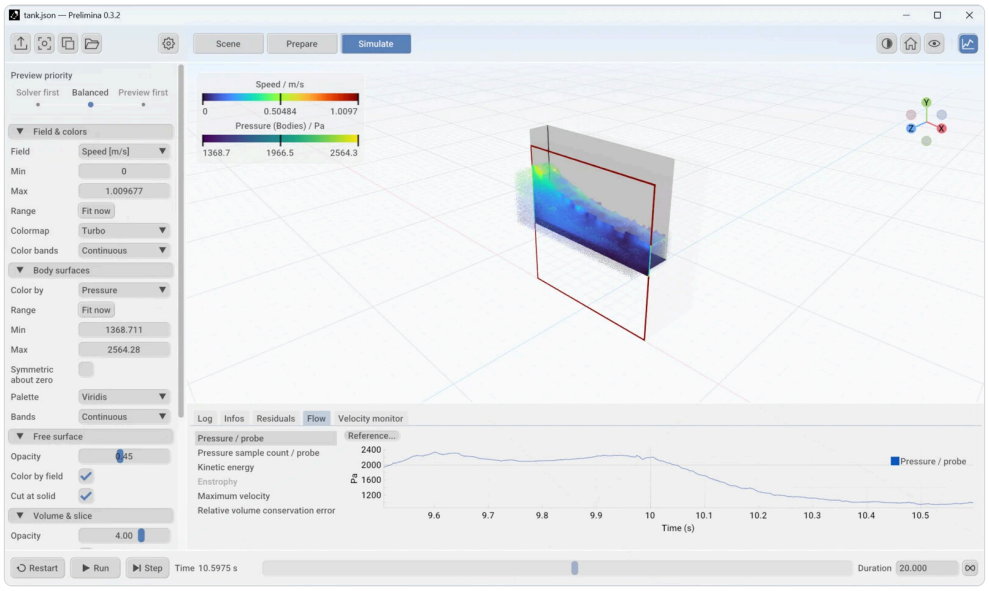


Figure 20. A slice through the tank at 10.6 s, coloured by speed. The Flow tab shows the probe pressure.

Field selects the quantity:

Field	What it shows
Speed	Flow speed in m/s
Grid pressure	Pressure on the grid. See the warning below.
Liquid level-set approximation	Distance from the liquid surface, in metres, negative inside the liquid
Grid density	Density on the grid, in kg/m³
Curvature	Curvature of the liquid surface, in 1/m

Min and **Max** set the colour range. **Fit now** fits it to the current values. To compare two moments or two runs, type the same range for both, otherwise the same colour means different values. **Colormap** and **Color bands** change the colours, and bands make levels easier to read.

> **Don't read pressure from the Grid pressure colours.** In 0.3.2 the viewport shows this field too high by a large factor. In the tank at a 0.02 m cell size, it showed −16,462 to 128,224 Pa, while the exported field for the same moment ran from −329 to 2,564 Pa, a factor of exactly 50. The shape of the field is still useful. For values, use a probe, **Body surfaces** or exported results.

Body surfaces colours a body by the pressure of the fluid next to it. Choose **Pressure** in **Color by**. It has its own range controls, and areas with no fluid beside them stay grey. **Wall shear stress** was greyed out for the tank, which does not simulate viscosity.

Section	Use it for
Free surface	Opacity of the liquid surface, and whether the field colours it. **Cut at solid** trims the surface at walls.
Volume & slice	Opacity of the field inside the liquid. **Liquid mask** limits it to the liquid.
Particles	Opacity and size of the particles

Move any **Opacity** slider to zero to hide that layer.

Look inside with a slice

1. Open **Volume & slice** and select **Clip & slice**. A coloured plane appears, and everything on one side of it is hidden.
2. Choose **X**, **Y**, **Z** or **Arbitrary** in **Plane**. **Arbitrary** adds a **Plane normal**.
3. Drag **Plane position**, from 0 to 1 across the flow.
4. Set **Contours** to draw lines of equal value on the slice.

Select **Plane handle** to move or rotate the plane in the viewport. The hidden side follows the camera, so orbit around to see the other side.

Read the diagnostics

Click the graph icon at the top right to open these tabs. A coloured dot on the icon means the Log has a new warning or error.

Tab	What it shows
Log	Solver messages, with a filter, a level list and **Copy**
Infos	Time, time step, size, speed and the output path
Residuals	How well each step was solved, against its target
Flow	Probe readings, kinetic energy and maximum velocity
Velocity monitor	Maximum and mean speed, and the count of invalid particles

The graphs in **Residuals** and **Flow** cover about the last second of the run, so they suit checking the flow as it runs, not reading a history. For a full record, use the probe file below.

Velocity monitor only checks whether the speeds are finite numbers. As it says on screen, it does not show that the solution is sound. Prepare a run has the checks that do, and explains why **Relative volume conservation error** can stay at zero when the volume is wrong.

Compare with a measurement

You can draw a measured curve over any graph in **Flow** or **Residuals**.

1. Select the graph, then click **Reference…** and choose a CSV file.
2. Give the file a header row. It needs a time column and a value column, with the values in the units shown on the graph.
3. Click **Align** if the curve needs adjusting.

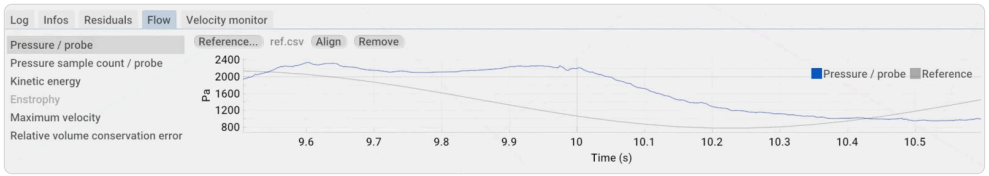


Figure 21. An invented sine curve in ref.csv, drawn over the probe pressure to show the overlay.

Align setting	What it does
Time column, Value column	Selects the columns to use
Time scale	Seconds per unit of the time column, for example the period T for data in t/T
Time offset	Shifts the curve in time, to line up the phases
Repeat every	Repeats the curve with this period. 0 shows it once.
Value scale	Multiplies the values

This is a quick visual check, and it only covers the last second of the run. For a real comparison, use the probe file and your own plotting tools. Note any offset you apply, because it is part of your result.

Save the numbers

Probes

Every probe writes to `results/probes` during the run, with no action from you. The first run writes `measurements.csv`, and later runs write `measurements-2.csv` and so on. A paused or stopped run keeps what it has written so far.

The file has one row per step for each series, so filter it by `series_id`. The tank probe has two series, the pressure and the number of particles it sampled.

Column	Meaning
`frame`, `time_s`, `dt_s`	Step number, time and time step
`series_id`	Which measurement, for example `pressure_probe.probe.pressure_pa`
`value`, `units`	The reading and its units
`availability`	`available` when the value is valid
`sample_count`	Particles behind this reading
`x_m`, `y_m`, `z_m`	Where the probe was

This Python script reads one series and prints its mean:

```
import csv

rows = [r for r in csv.DictReader(open("measurements.csv"))
        if r["series_id"] == "pressure_probe.probe.pressure_pa"
        and r["availability"] == "available"]
t = [float(r["time_s"]) for r in rows]
p = [float(r["value"]) for r in rows]
print(len(p), "samples, mean", sum(p) / len(p), "Pa")
```

The first sample of a run can be 0, because the probe has no pressure before the first step. Leave it out of averages.

Fields at one moment

Write results, the first icon at the top left, saves the current state.

1. Click **Write results**. The run pauses.
2. Check the **Folder**, which is the `results` folder by default.
3. Click **Write now**, then **Close**.

Each click adds a `step_NNNNNN` folder and an entry in `results.vtm.series`. One frame of the tank took about 10 MB:

File	What it holds
`particles.vtp`	Particle positions and velocities
`free_surface.vtp`	The liquid surface as a mesh
`grid/*.vti`	Grid fields, including `pressure` and `velocity`
`results.vtm.series`	The list of saved moments. Open this one.

A folder can only take later moments of the same run. After a **Restart**, choose a new folder.

Fields over time

The application saves fields only when you click. **Record results**, which saves them at regular intervals, is a locked beta feature in 0.3.2. To record a series without unlocking it, use `prelimina_cli`, which is beside the Prelimina program:

```
prelimina_cli tank.json --backend vulkan --output metrics.csv --vtk-dir fields --
vtk-every-time 0.5 --vtk-product volume --vtk-field pressure --vtk-field velocity
```

This writes the grid fields every 0.5 s of simulated time, and `--duration` overrides the end time. Each run records one product, and FLIP has `volume`, `particles` and `free_surface`. Without `--vtk-product` you get particles and the free surface. Run the same setup again for another product.

The command line has no `results/probes` folder. Probe values are in `metrics.csv` instead, as `result_pressure_probe_...` columns, with `;` as the separator and several hundred columns in all.

Start small. The tank with particles and surface took about 1.4 MB per frame, and a fine grid or a short interval can fill a disk quickly.

Pictures

Use **Screenshot** to save the viewport as a PNG, and **Copy image** to put it on the clipboard. Neither includes the panels. The folder icon opens the results folder.

Open results in ParaView

1. In ParaView, choose **File > Open** and select `results.vtm.series`.
2. Click **Apply**. The time controls step through the saved moments.
3. Use the block list to choose particles, grid or free surface.

Use this	Not this
Grid `pressure`, in Pa	Particle `pressure`, which is 0 in every particle in 0.3.2
Grid `velocity`, where `velocity_valid` is 1	Grid values outside the liquid

Exported grid pressure is in Pa, and its maximum matched **Body surfaces** at the same moment, so ParaView is the place to measure a field.

If a run stops

The bar shows **Initialization failed** or the run stops, and a message appears in the viewport and in the **Log**. Copy it with **Copy** if you need to contact support.

Message contains	Cause	What to do
`exceeding the particle capacity`	The initial liquid needs more particles than allowed	Raise **Particle capacity**. See Particles.
`density projection candidate validation failed`	Usually a time step that is too long	Use the recommended settings

A run that finishes can still be wrong, so a clean Log is not enough. See Check a setup before a long run.

Next steps

- See measured cases in Applications.
- See what each feature can do today in Capabilities.