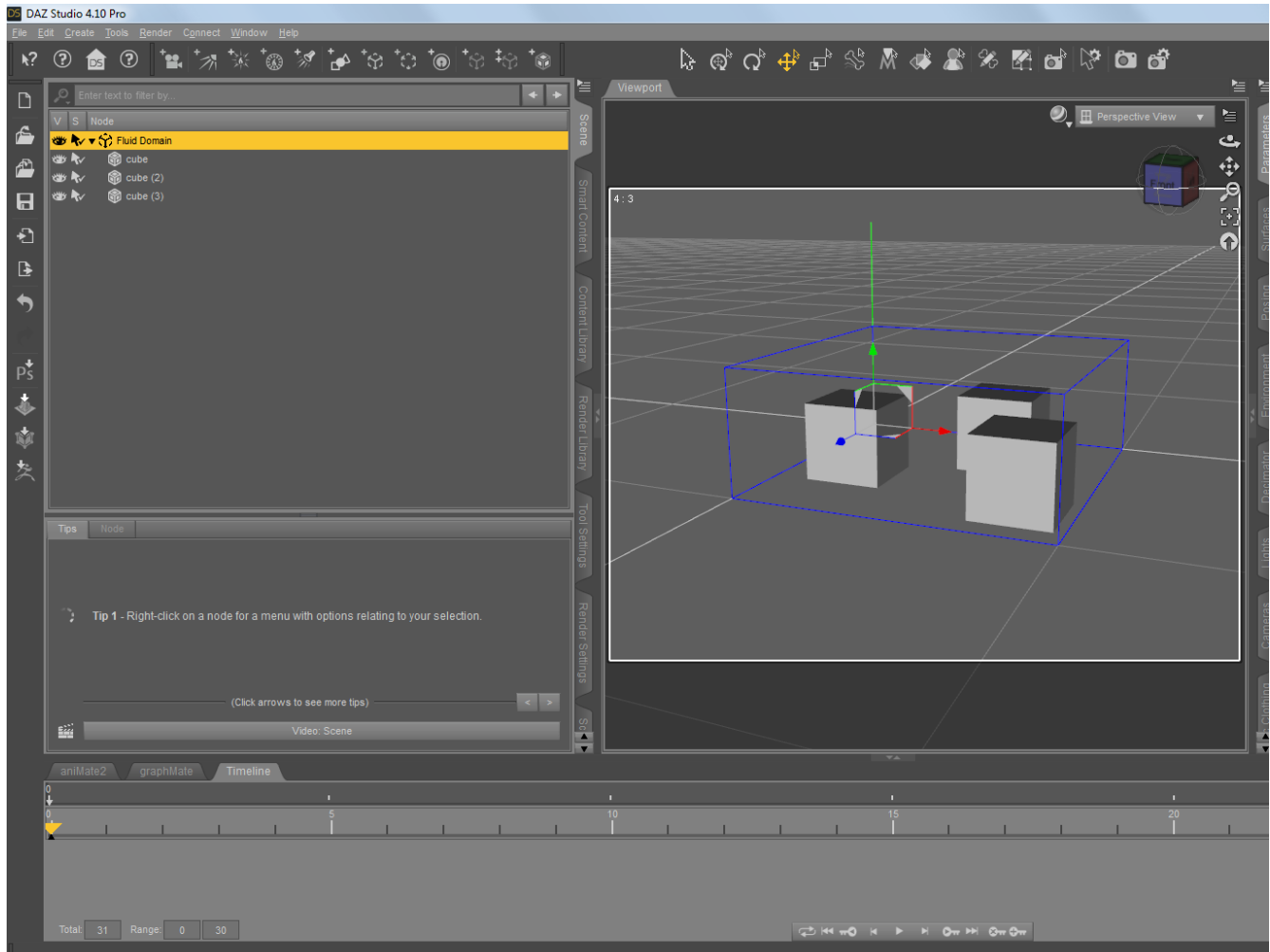


FLUIDOS for DAZ Studio
Recipes

OBSTACLES

1. Create a new Fluidos Domain. Set Size Y to 40.
2. Create three cubes of 0.25 m of size (of whatever size you want as the cubes can be positioned inside the Domain without overlapping)
3. Arrange the cubes inside the domain and parented them to it.



4. Set ON “This is a fluid mass” to one of the cubes. Let the others as the are (obstacles, by default).
5. Set the properties of Fluidos Domain as this:
 - Number of frames: 30
 - Cell size: 2
 - Subdivision level: 2
6. Select a suitable folder for saving the bakefiles.
7. Save your scene.
8. Run the simulation.
9. Create a New Fluidos Mesher. Put a keyframe in the timeline at 0 frame and then go to 30 frame, set Completion to 100 % (At 0 frame Completion should be 0 % by now).
10. Copy the Fluidos domain (Ctrl+C) to the Mesher (the scale must be locked in the Mesher). This form, the folder will be copied to the Mesher (if you prefer, you can choose them manually).

11. Enable the Mesher. Set off the preview.
12. Set the first cube (the fluid) to invisible.
13. Explore the animation.

If you want to put a shader on the fluid, apply a preset on the Mesher (or edit the default).

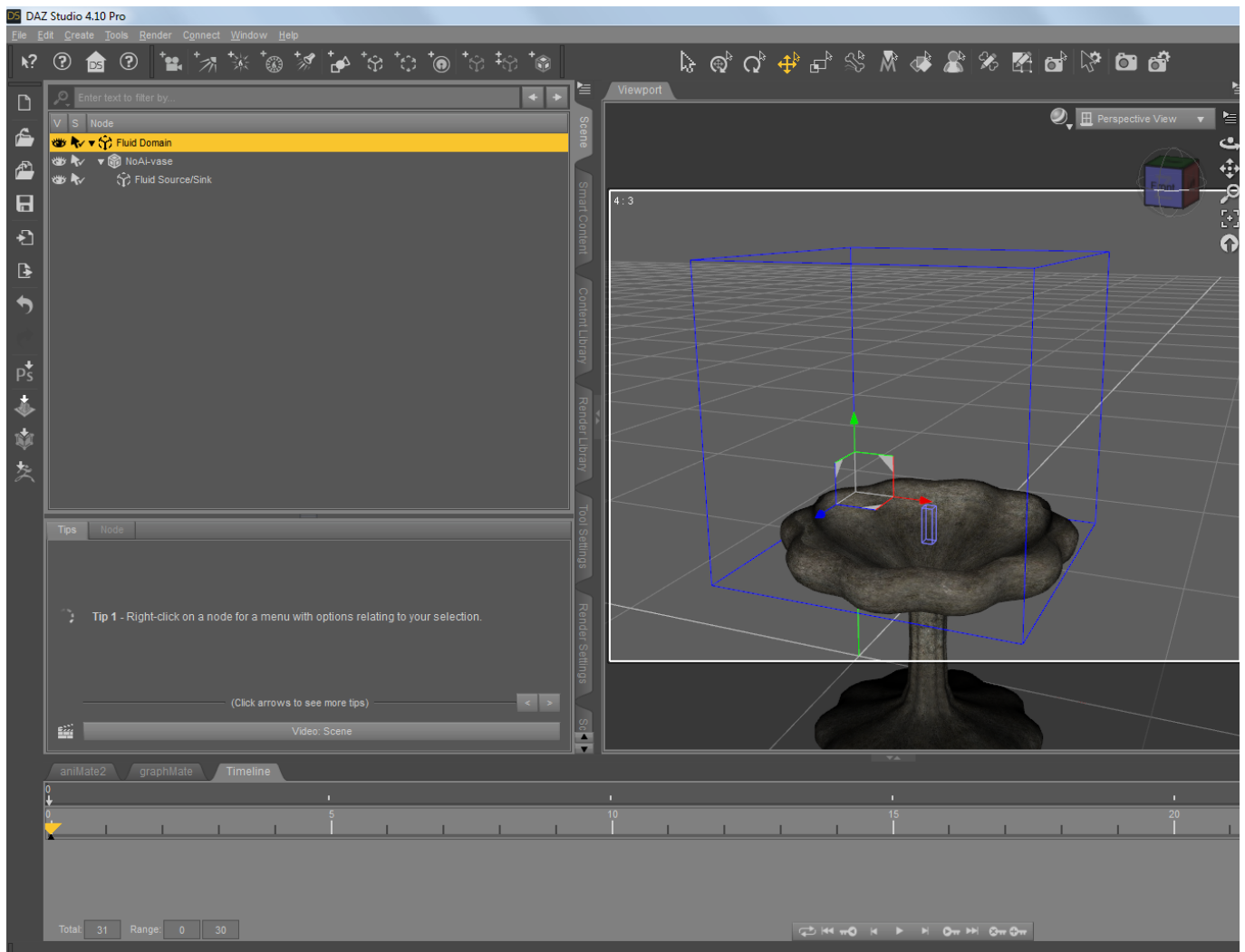
To experiment:

Set OFF the Mesher, enable surface force in a solid cube, redo the simulation and see the differences (positive values are for attracting forces; negative, for repelling).

Now enable viscosity. Try different values of viscosity. See the effect of increasing or decreasing the Marker Particles scale.

THE FOUNTAIN

1. Put a NoAi-Vase (from Genesis Starter Essentials) in scene.
2. Create a new Fluidos Domain. Reduce the size to X=80, Y=80, Z=80. The less wasted space in the domain, the faster the simulation.
3. Create a new Fluidos Source/Sink. Set the size to X=2, Y=10, Z=2. This is a skinny but tall source. Set the velocities to X=0, Y=350 and Z = 0.
4. Translate the Domain and the source to the appropriate position in the scene. Parent all to the Domain.



5. Set the cell size in the Domain to 1.25, the number of frames to 30. You can use a larger cell size to fast tests, or lower size to more detailed results.
6. Set ON the Diffuse particles (whitewater).
7. Select a suitable folder for saving the bakefiles.
8. Save the scene.
9. Run the simulation.
10. Create a New Fluidos Mesher. Put a keyframe in the timeline at 0 frame and then go to 30 frame, set Completion to 100 % (At 0 frame Completion should be 0 % by now).

11. Copy the Fluidos Domain (Ctrl+C) to the Mesher (the scale must be locked in the Mesher).
12. Duplicate the Mesher. In the duplicate, set ON Include diffuse particles, and Particles size to 2.
13. Now, enable the two Meshers. Set Preview OFF in the first Mesher.
14. Explore the simulation along time.

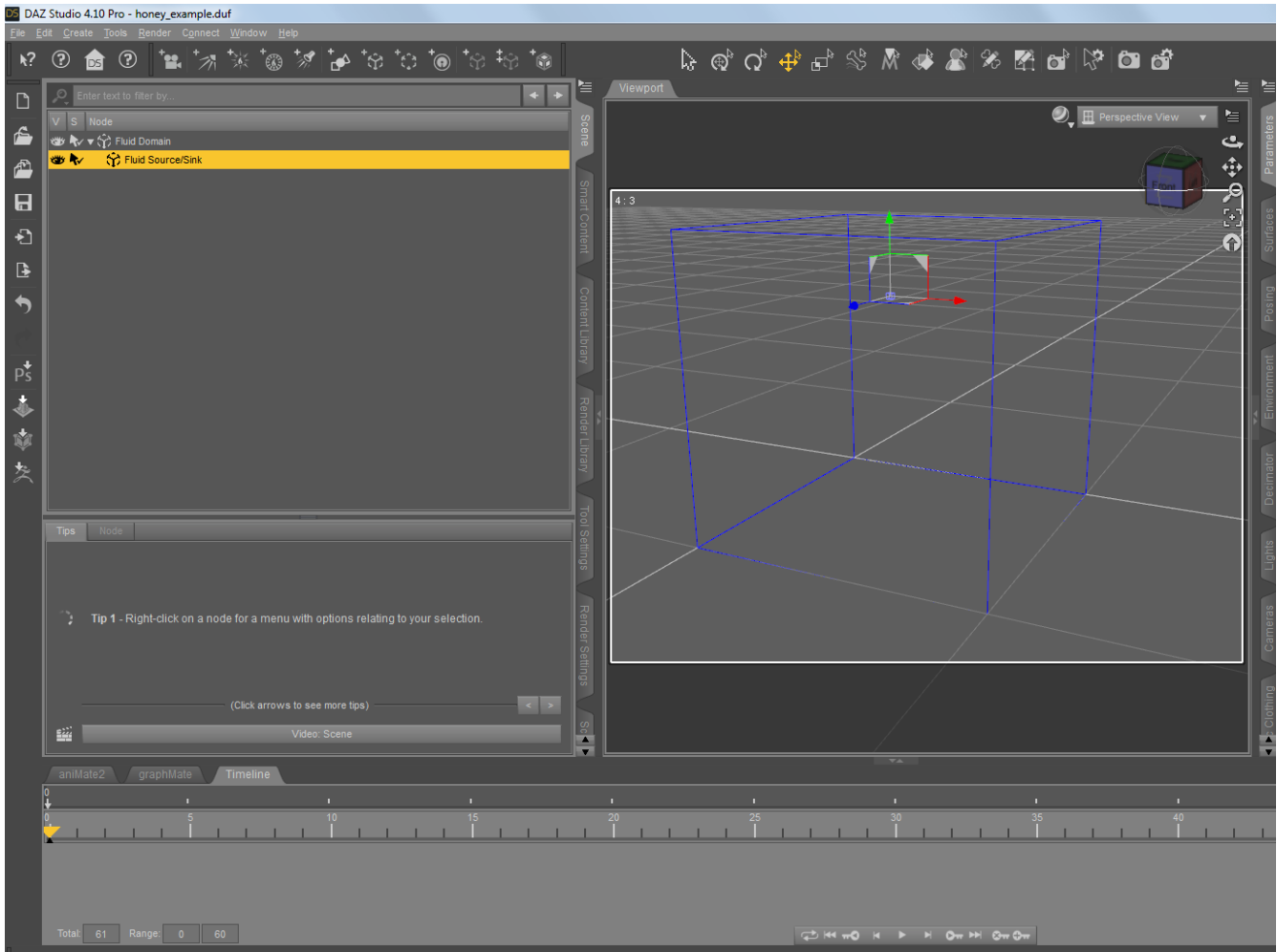
To experiment:

Try to change some of the parameters to see the results in a new simulation. It is recommended to set OFF the Mesher before redo a simulation to avoid delays due to the automatic redrawing of the fluid at each frame (the plugin need to pass through all the specified frames in the timeline during running simulation).

Try to set ON only the second Mesher (the diffuse particles one) to see the effect.

HONEY

1. Create a new Fluidos Domain.
2. Create a new Fluidos Source/Sink. Set its size to X=2, Y=2 and z=2. Set X or Z velocity to 3. Translate this source to some upper position inside the Fluid domain.
3. Parent the source to the Domain.



4. In Fluidos domain parameters, set:
 - Number of frames: 60
 - Cell size: 2
 - Subdivision level: 2
 - Enable viscosity: ON
 - Viscosity: 200
 - Marker Particles scale: 4
5. Select a suitable folder for saving the bakefiles.
6. Save your scene.
7. Run the simulation.
8. Create a New Fluidos Mesher. Put a keyframe in the timeline at 0 frame and then go to 30 frame, set Completion to 100 % (At 0 frame Completion should be 0 % by now).

9. Set the previously selected (step 5) Baked files folder in the Mesher.
10. Enable the Mesher. Set off the preview.
11. Explore the animation.

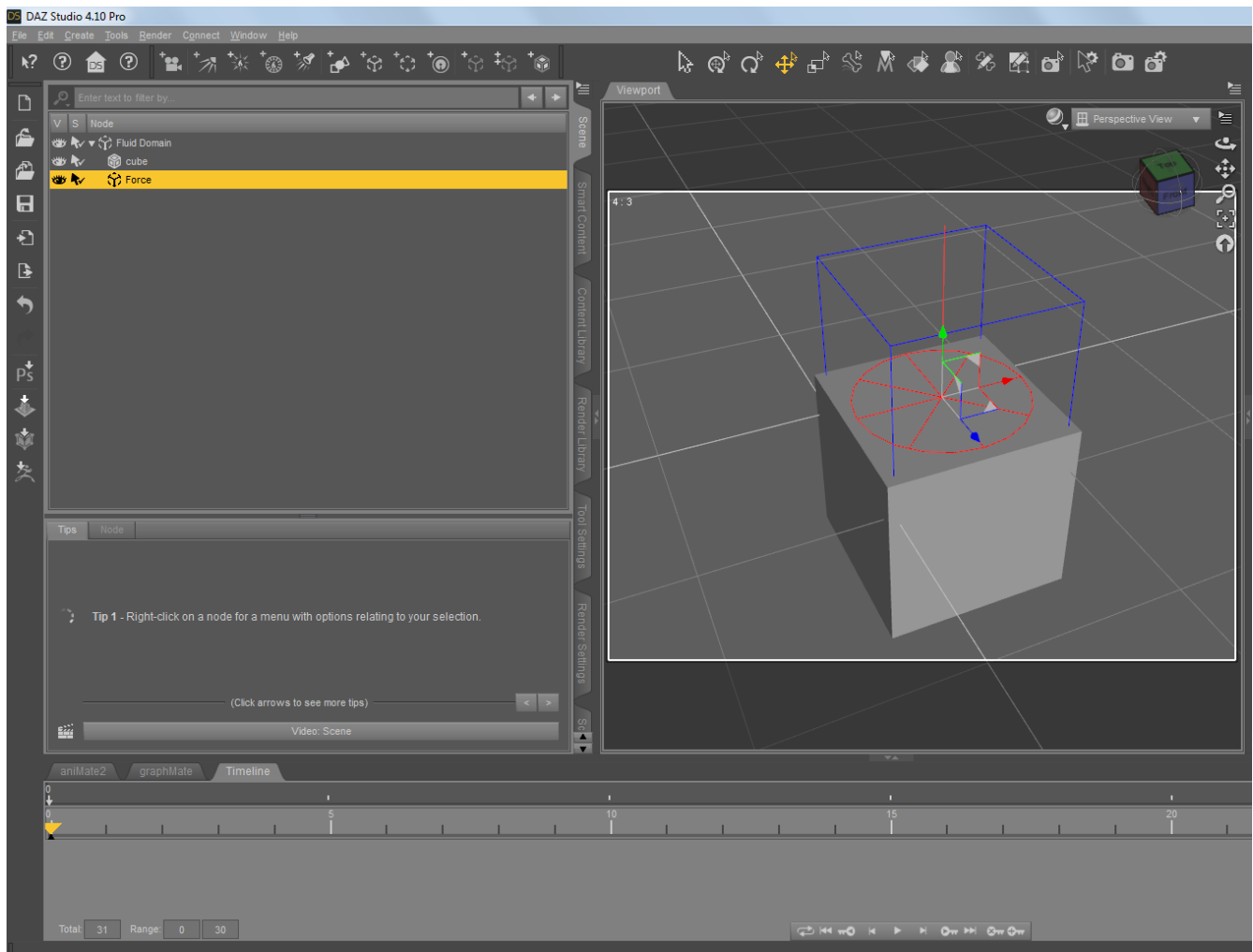
To experiment:

Try different values of viscosity and redo simulation.

Decrease or increase the Marker Particles size to see the effect on the simulation.

WHIRPOOL

1. Create a new Fluidos Domain.
2. Create a cube of 1.1 m size. Move it to Domain so the cube could fills the lower fourth part. Set ON This is a fluid mass.
3. Create a New Fluidos force. Set Force type to Torque. Set the strength to 30. Translate the force inside the domain. Let the center of the circle at the domain XZ plane center (see figure).
4. Parent the cube and the force to the domain.



5. In Fluidos domain parameters set:
 - Number of frames: 30
 - Cell size: 2
 - Subdivision level: 1
6. Select a suitable folder for saving the bakefiles.
7. Save your scene.
8. Run the simulation.
9. Create a New Fluidos Mesher. Put a keyframe in the timeline at 0 frame and then go to 30 frame, set Completion to 100 % (At 0 frame Completion should be 0 % by now).
10. Set the previously selected (step 6) Baked files folder in the Mesher.

11. Enable the Mesher. Set off the preview.
12. Explore the animation.

To experiment:

Increase the subdivision level and redo simulation to see the effect.

Rotate the torque force around the X axis (or Z axis) and run the simulation again.

Change the type of force to point force and see the effect after running the simulation.

Set ON the Include diffuse particles in the Fluidos domain. Redo the simulation. Create another Mesher to visualize the diffuse particles.