**ETH**
Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

**Computational Statistical Physics**
**Exercise sheet 07**

FS 2020
Oded Zilberberg

**Exercise 1.   Interacting particles in a box**

*Goal: This week's exercise is about simulating a system of $N$ interacting particles in a box using the Verlet scheme.*

**Note:** The code of this exercise will be the basis for further improvements and modifications in the following exercises. Therefore, please make sure that it is easily readable and well-structured.

Consider a system of $N$ point particles (in a box of length $L$ with periodic boundary conditions) that interact via the Lennard-Jones potential

$$V(r) = 4 \left[ \left( \frac{1}{r} \right)^{12} - \left( \frac{1}{r} \right)^{6} \right]$$

with each other. The total Hamiltonian of the system is given by

$$H = \frac{1}{2} \sum_i \boldsymbol{v}_i^2 + \frac{1}{2} \sum_{i \neq j} V(|\boldsymbol{r}_i - \boldsymbol{r}_j|).$$

**Task 1:** Simulate the system by using the Verlet scheme.

*Hint:   Use sufficiently small time steps and a cutoff length $r_c = 2.5$ for the Lennard-Jones potential.*

**Task 2:** Investigate the influence of the choice of the time step size.

**Task 3:** Measure the energy of the system as well as its fluctuations. Is the energy conserved?

---

**Visualization of the results**: For visualizing the particles you have different options. You can simply use a graphical tool (e.g. matplotlib or gnuplot) which reads in the coordinates for a given time step. Another possibility would be to use either vmd ([https://www.ks.uiuc.edu/Research/vmd/](https://www.ks.uiuc.edu/Research/vmd/)) or paraview ([https://www.paraview.org/](https://www.paraview.org/)) to visualize the dynamics and create movies.

If you want to use paraview then the following information might help you to get started. To be able to visualize your simulation you may use the provided `vtktools.py` and `convert_to_paraview.py` to bring your computed data into an appropriate shape for paraview.

Once your simulation is finished you can go over to paraview and follow the steps below:

1. Load the data files.

   - Open paraview.
   - Go to "File" → "Open" and choose the group `MD..vtu` in the folder `simu`.
   - Click "Apply" in the "Properties" window.

2. Display the particles.

- Click "MD\*" in the "Pipeline Browser" window.
- Find the "Glyph" symbol in the toolbar and click it.
- In the "Properties" window set "Glyph Type" → "Sphere", "Vectors" → "Points", "Scale Mode" → "Scalar", "Scale Factor" → "0.5", "Glyph Mode" → "All Points" and click "Apply".

3. Add the forces.

- Click again "MD\*" in the "Pipeline Browser" window and add another "Glyph".
- In the "Properties" window set "Glyph Type" → "Arrow", "Vectors" → "forces", "Scale Mode" → "Scalar", "Scale Factor" → "0.75", "Glyph Mode" → "All Points" and click "Apply".

4. Add a box.

- Go to "Sources" → "Box" and adjust it properly. E.g. in the case where $L = 10$ set "X Length" → "10", "Y Length" → "10", "Z Length" → "10" and "Center" → "5 5 5". Click "Apply". Set "Opacity" → "0.2". Click "Apply".

5. Click the "Play" symbol in the toolbar. (Note: You might need to zoom out and/or move around in the coordinate system to see something.)

To save the simulation movie simply go to "File" → "Save Animation" and follow the dialogue.