What do we define as a multiphase flow? We deem so every flow in which the interaction between at least two media in different physical states, i.e. solid, liquid or gas, occurs. Bubbles of carbon dioxide rising in a glass of champagne, rain or blizzards are everyday examples of those phenomena. Such flows are also important for the industry: lubrication of machine parts (e.g. bearing chambers), boiling and steam generation to name a few. Research of these flows allows for more accurate prediction of natural phenomena (everyone loves correct weather forecast, doesn't he?), as well as more efficient and safer processes in chemical, food, oil and gas industries and many more. We have two scientific tools to choose from: experiment and numerical modelling. Due to the complexity of multiphase flows, experimental studies are usually hard and expensive. Numerical modelling, or to put it more bluntly computer simulations, are much cheaper alternative, allowing for nearly unlimited choice of physical properties of gases and liquids, and geometries of investigated devices.

The objective of the project is development and testing of one particular method used for computer simulations, called Smoothed Particle Hydrodynamics (SPH). SPH belongs to the group of the so-called meshless approaches. This means that continuous medium (gas or liquid) is treated as a set of many particles, for which equations of motion are solved. SPH is commonly used in entertainment industry for special effects, like tsunami waves flooding New York city, in movies or video games. The proposed project, however, is a scientific one. So why exactly SPH? This method can fairly well deal with significant density ratios (water is around thousand-fold denser than air!) and, what is more important for us, it is exceptionally well suited for the so-called massively parallelised computations. Numerical simulations, despite generally lower cost than experiments, require access to powerful computers. Such machines are equipped with multiple processors, which allows one to split computational tasks to smaller ones, solved simultaneously, hence in parallel. Instead of processors (8-16 tasks at the same time) Graphic Processing Units (GPUs) can be used (hundreds or even thousands of parallel tasks). The first aim of the project is the development of software basing on the SPH method, tailored and optimised for execution on computers equipped with multiple (at least four) GPUs.

This software will then serve as a tool for modelling of two-phase flows in ducts. Depending on the volume ratios between liquid and gas, such flows can contain various structures, like (i) groups of small gas bubbles moving in a liquid, (ii) completely chaotic, churning structures or (iii) annular systems (liquid covering the walls, gas filling the centreline of the channel). Despite simple geometries, the modelling of this type of flows is very difficult – most of the available numerical methods have their limitations or require tremendous computational power. What is more, we are interested in fully resolved simulations, i.e. computations in which accuracy is high enough to reproduce even the smallest flow structures. The SPH method has high chance of becoming a useful tool for engineers and scientists investigating multiphase systems. SPH can decently deal with complex interfacial flows, yet its biggest drawback is relatively high computational cost. The opportunity to perform computations on the GPUs (much cheaper than standard processors) nullifies this flaw. The lion's share of the project will be dedicated to assessment of agreement of the results obtained from calculations with experimental ones. Confirmation of the SPH effectiveness in this field will be important for future applications to real technical problems.

Another challenge that will be undertaken in the project is the well-known SPH problem of unphysical degradation of the liquid-gas interface (for simplicity one can think of it as a thin membrane between phases). Every rose has its thorns, but luckily, we have an idea how to dull them. Due to the representation of fluid as small particles, in SPH we can observe undesired mixing of different phases. In nature, presence of the surface tension forces results in immiscibility of separate phases, e.g. drops of oil in water. In SPH, where physical model is necessarily simplified, we can observe separation of single particles. This is an incorrect and purely numerical effect. To prevent it, a small repulsive force is sometimes added, which repels phases a bit from each other. Such approaches are flawed, due to their artificial character and negative influence on the physics of the flow, e.g. premature break-up of bubbles rising in a liquid. In the project we will develop a novel method in which such single particles will be treated as small droplets/bubbles, modelled with separate, physically sensible equations. Such remedy will allow us to obtain more accurate results, free of errors related to the artificial correction terms.

To summarise, in the described project we will focus on the development of reliable and efficient numerical approach, dedicated for computations of the gas-liquid flows. The novelty of GPU usage and physically-sound model to treat undesired numerical effects should catch attention and extend a helping hand towards scientists and engineers, for whom multiphase flows are bread-and-butter (and not really a piece of cake).