Morpheus Fluid

ACCURATE MESHLESS COMPUTATIONAL FLUID DYNAMICS

Dr. Linghan Li and **Prof. Justin Read*,** University of Surrey, linghan.li@surrey.ac.uk

Let 's talk!

Scan:
https://morpheusfluid.com/

PROBLEM

Computational fluid dynamics (CFD) is a numerical method that allows us to digitally model wind flow around cars, the impact of water on flood defence barriers, the flow of ink through an inkjet printer nozzle, and even the flow of blood through the body. CFD allows us to build "digital twins" that accelerate design, leading to more efficient and more effective products.

However, standard CFD methods rely on "meshes" (see Fig. 1, left) that, for complex problems, can take 3-5 months of expert time to set up. This limits access to the benefits of CFD to companies with significant resources.

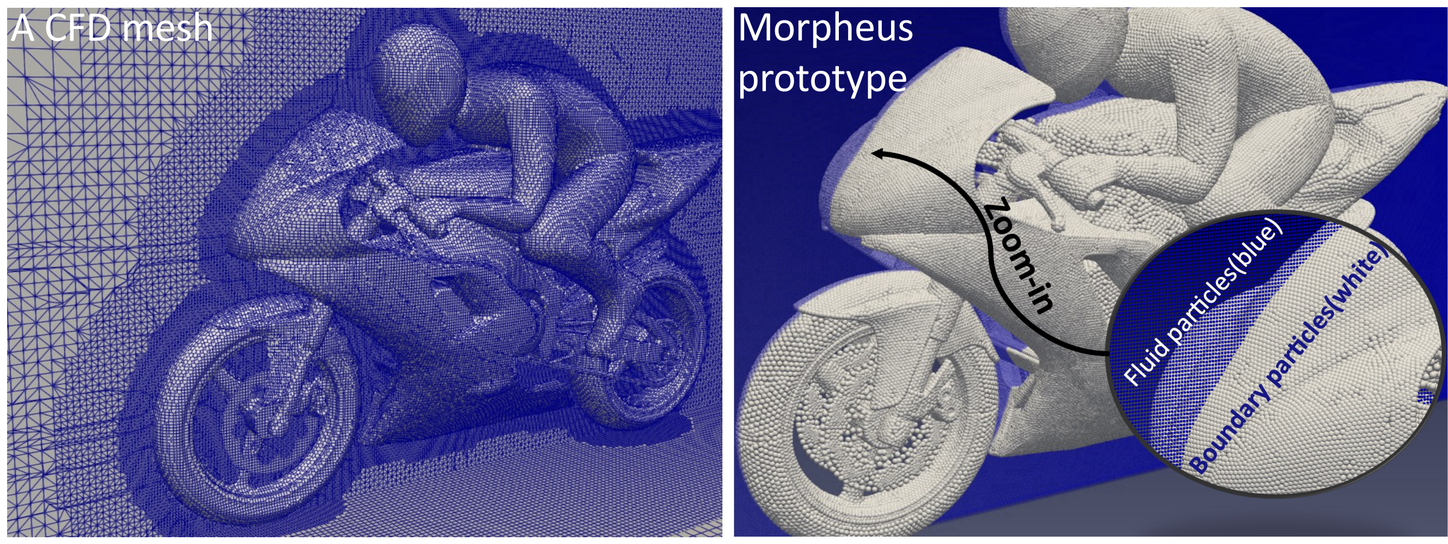


Figure 1: *Setting up a CFD model of air flow around a motorbike.* ***Left:*** *standard CFD methods use "meshes" that take expert time to set up(>10 hours overall in this simple case).* ***Right:*** *Morpheus uses "particles" instead that move with the flow, completely removing the mesh set-up time.*

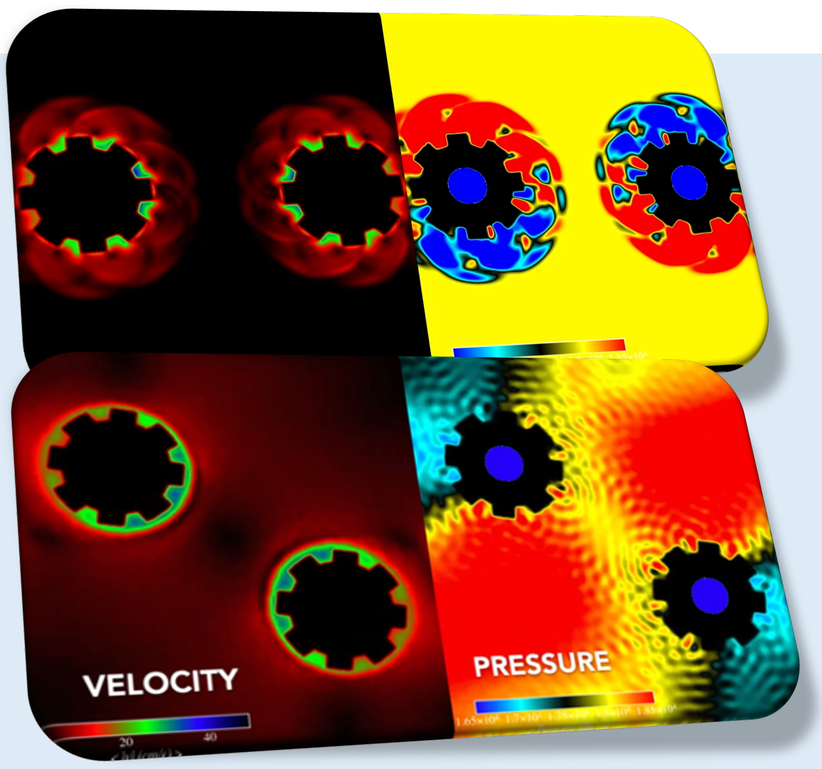


Figure 2: *Morpheus can model time-evolving geometries, like this simulation of two gears rotating in a turbulent water tank, as easily as if the gears were not there.*

CHALLENGE

One approach to solving the problem of mesh-generation is to eliminate the need for a mesh by using "particles" that co-move with the flow instead (see Fig. 1, right). This also has the advantage that moving/flexing walls, barriers and gears can all be modelled with ease (see Fig. 2).

However, current state-of-the-art "mesh-free" methods are inaccurate and unable to capture important physical phenomenon like turbulence. This has limited their usage, so far, to more niche problems for which accuracy is not so important.

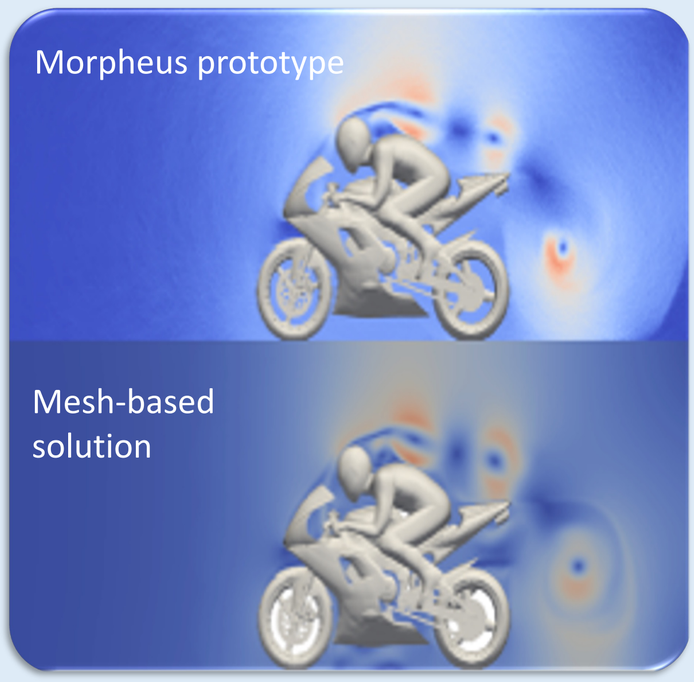
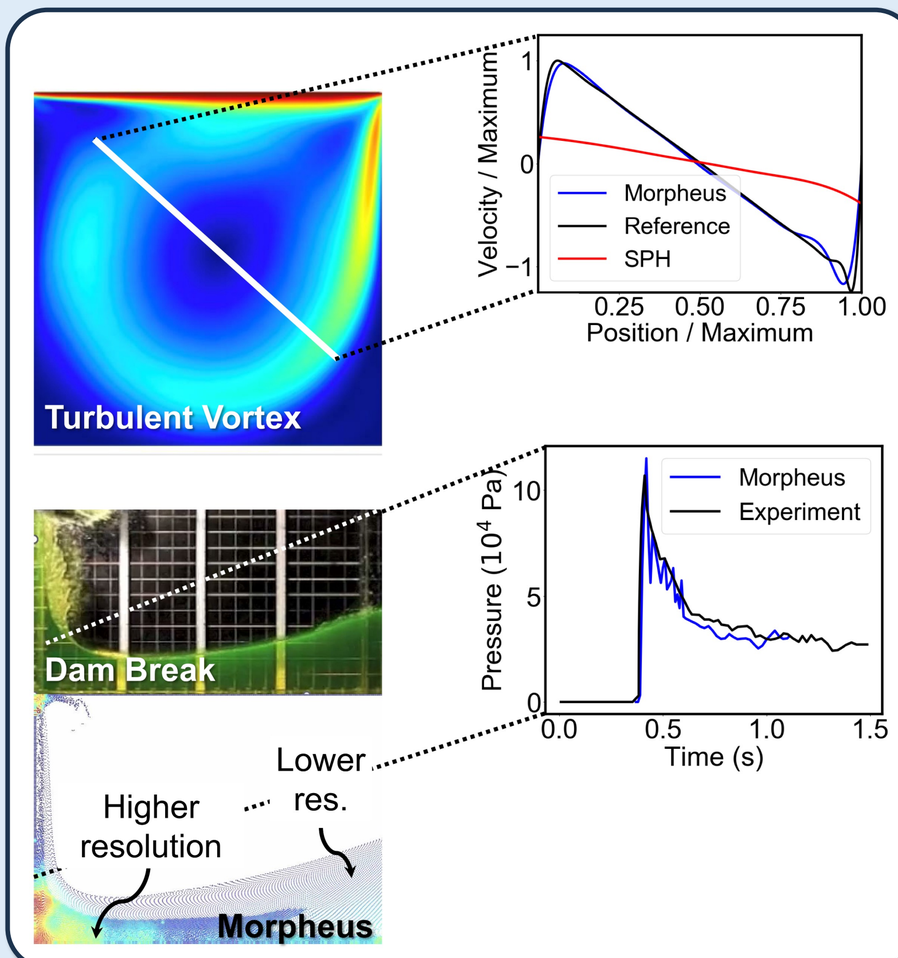


Figure 3: *Modelling the air flow around a motorbike. The contours show regions of high (red) and low (blue) velocity. Unlike previous meshless CFD methods, Morpheus (top) is as accurate as mesh-based methods (bottom).*



SOLUTION

We have developed a new "mesh-free" CFD algorithm – Morpheus – that eliminates the need for a mesh but is as accurate as mesh-based CFD methods (Fig. 3 and Fig. 4). This will significantly reduce the cost to run CFD calculations, opening up CFD to a whole new market of non-expert users. Morpheus will boost productivity, improve safety, and raise the efficiency of designs in industries as wide-ranging as automotive, aerospace, environment and energy.

Figure 4: *Morpheus is meshless, but as accurate as state-of-the-art mesh-based CFD methods.* ***Top:*** *Morpheus models a turbulent vortex with a Reynolds number of 10,000. The contour plot shows a visualization of the vortex, where the contours show the magnitude of the fluid velocity. The graph shows that Morpheus (blue) gives an excellent match to the reference solution (black), whereas a previous state-of-the-art meshless method (SPH; red) fails.* ***Bottom:*** *Morpheus accurately models a "dam break" test where water impacts a wall. Here, we also demonstrate Morpheus's ability to use multi-resolution particles to "zoom-in" on regions of interest.*

IMPACT

Morpheus offers the following key advantages over standard CFD methods:

- Elimination of the expensive pre-simulation mesh set-up phase;
- Insightful simulation analysis through automatic time-tracking of fluid elements;
- Ability for non-expert users to harness state-of-the-art CFD tools, for the first time;
- Modelling of free surface flows, complex geometries, and moving parts, all with ease;
- Efficient modelling of high velocity flows.

Simulating an aeroplane landing on a wet runway with our Morpheus prototype

STEM for BRITAIN
UNIVERSITY OF SURREY
UKRI Engineering and Physical Sciences Research Council
UKRI Science and Technology Facilities Council
UKRI Innovate UK