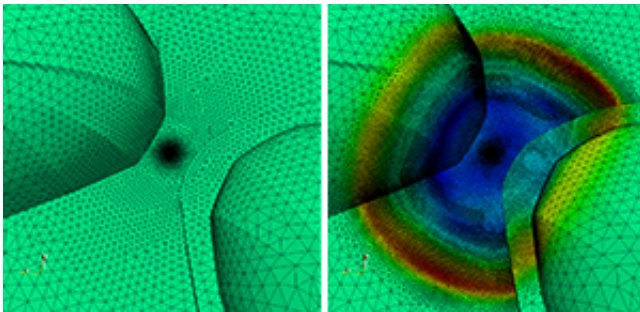


Fluid dynamics: Resolving shockwaves more accurately

July 17 2013



Simulations of shockwaves in fluids as they initiate (left) and propagate (right), using a specially tuned computational mesh. Credit: 2013 A*STAR Institute of High Performance Computing

A new computational scheme enables more stable simulations of shockwaves in fluids and may be scalable for large engineering designs.

Vinh-Tan Nguyen and co-workers at the A*STAR Institute of High Performance Computing in Singapore have developed a more robust and efficient way to simulate [shockwaves](#) under various flow scenarios¹. Previous techniques for shockwave simulation are specific to particular flow problems, whereas this new method is applicable to shockwaves in any high-speed flow scenario, for example in aerodynamics or explosions.

A shockwave is generated when a discontinuous change in fluid

properties follows an abrupt increase in the pressure, temperature and density of the flow. "Strong and unsteady shockwaves can produce [oscillations](#), which affect the stability of numerical solutions in the three-dimensional (3D) computational domain," explains Nguyen. "The main aim of our technique is to resolve the front of a shockwave while preserving the overall accuracy of the simulation."

In [computational fluid dynamics](#), flows can be simulated with different levels of accuracy—a low-order approximation is based on a hypothesis, whereas a high-order approximation is one that is closest to reality, or the 'finest-tuned' approximation. Simulation accuracy is maintained by using as high-order approximation as possible, as well as by altering the resolution of the 3D computational mesh—a grid of interconnected data points that covers the spatial area of the flow.

"Simulating flows using high-order approximations triggers oscillations, which cause miscalculations at the front of shock waves where the flow is discontinuous," explains Nguyen. "It therefore becomes counterproductive to have high-order approximations in place right across shock regions."

To overcome this problem, Nguyen and his team placed a shockwave sensor within the flow to identify high-gradient shockwave fronts as they appeared. They then applied shock capturing schemes to resolve the fronts by reducing the approximation order in those specific regions.

Finally, the researchers increased the spatial resolution of the computational mesh in the localized shock areas to compensate for the lower-order approximations (see image). The 3D mesh is also programmed to rebuild itself following contact with a shockwave.

"With precise detection through the shockwave sensor we can apply the right capturing scheme to treat each shockwave, regardless of its

strength," explains Nguyen. "Our mesh adaptation procedure then simultaneously refines the mesh in shockwave regions and coarsens it in areas of least change, reducing computational costs significantly."

In addition to its potential application in [aerodynamics](#) and blast analysis, the researchers believe that this scheme may be useful for simulating the interface between air and water, with huge potential for marine and offshore applications.

More information: *Computers & Fluids* 70, 126–135 (2012). [doi: 10.1016/j.compfluid.2012.09.011](#)

Provided by Agency for Science, Technology and Research (A*STAR), Singapore

Citation: Fluid dynamics: Resolving shockwaves more accurately (2013, July 17) retrieved 25 April 2024 from <https://phys.org/news/2013-07-fluid-dynamics-shockwaves-accurately.html>

<p>This document is subject to copyright. Apart from any fair dealing for the purpose of private study or research, no part may be reproduced without the written permission. The content is provided for information purposes only.</p>
--