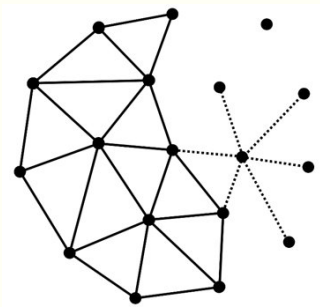
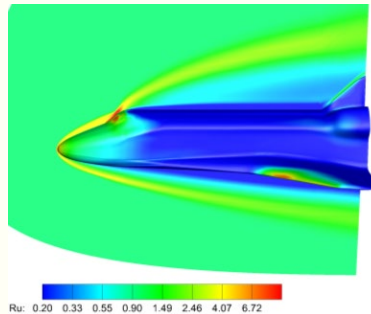


Robust and accurate method for solving computational fluid dynamics (CFD) and other complex problems

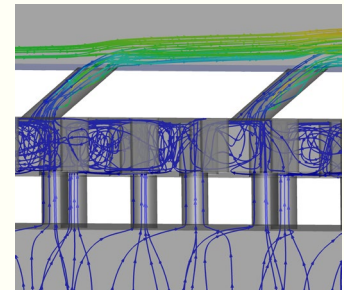
Applications: Aviation, Automotive, Power Generation, Engineering, Blood Flow



Unified mesh/meshfree CFD
with mixing of nodes and points



Simulation of the
space shuttle



Double-wall turbine
cooling

A unified mesh/meshfree method for computational fluid dynamics without complex mesh treatment and failures at mesh/meshfree boundaries

Features	Benefits
Combines the geometric flexibility of meshfree CFD with the physical accuracy of mesh methods	As general, automated, efficient, and flexible as meshfree methods but more stable and accurate
Avoids poor accuracy of previous meshfree methods	High accuracy simulation of physical systems
Automatic mesh generation without manual adjustment	Avoids need for time consuming input by experts
Inherently avoids meshing failures (negative-volume elements; low-quality elements)	Robust simulation without meshing failures and artifacts
Suits arbitrarily complicated configurations even with moving boundaries	Straight forward simulation of complex geometries including moving surfaces
Can be applied to other physical problems governed by the complex advection-diffusion PDEs	Could be applied to fluid mechanics, solid mechanics, electromagnetism, etc.

Patented and available for

- Licensing

Learn more:

enquiries@innovation.ox.ac.uk

Browse more opportunities:

www.innovation.ox.ac.uk