UNIVERSITY OF CAMBRIDGE

Mesh generation for continuum modelling (Part 1)

Dr. Nandan Gokhale, 2023

MPhil in Scientific Computing
Centre for Scientific Computing
Cavendish Laboratory

(with thanks to Dr. L. Michael, and Dr. N. Nikiforakis)

Structure of this lecture

- Introduction and fundamentals
- Grid types
- Cartesian cut cell methods (fundamentals)
- Summary and outlook

UNIVERSITY OF CAMBRIDGE

Introduction and fundamentals

Continuum discretisation

https://www.contourheating.co.uk/hubfs/DSC_0873.jpg

- Problem: To model the flow of aerosols in this room.
- Treat the circulating air as a continuum.
- **Discretisation** critical when translating a continuum mathematical problem into a computational framework, which has finite memory and compute power.

UNIVERSITY OF CAMBRIDGE

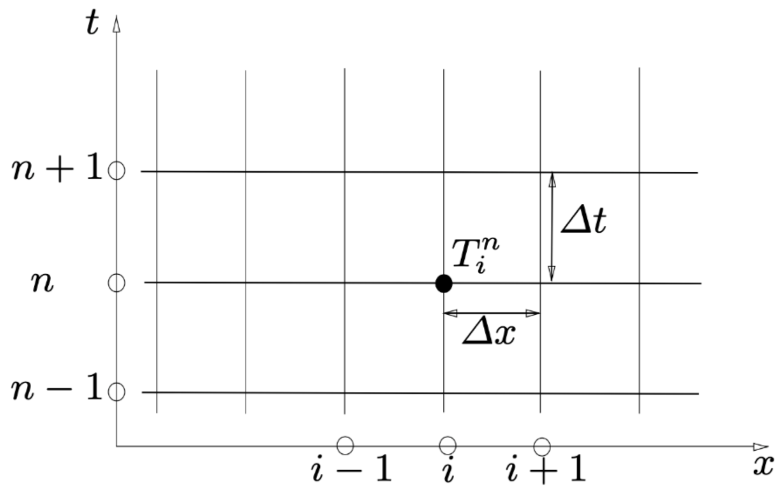
Continuum discretisation

Definition: *Discretization is the process of approximating a problem defined on a continuum by a 'discretized' problem involving a finite set of values.*

- For example, recall the parabolic 1D heat equation:

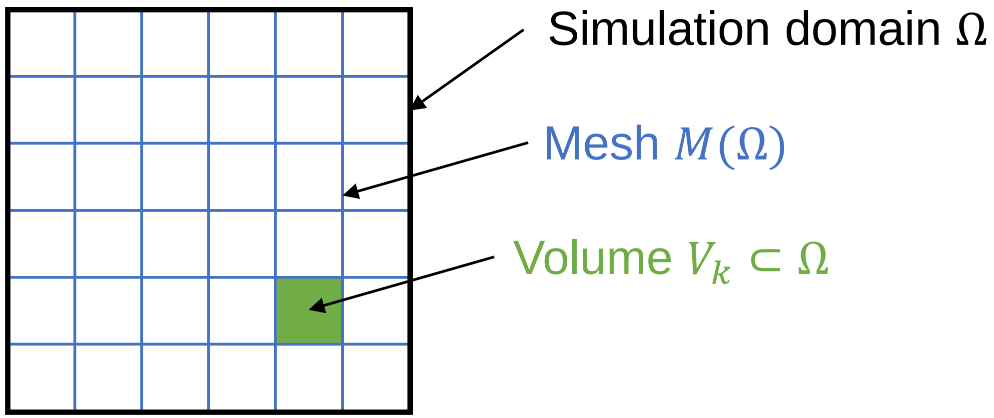
$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

Spatial and Temporal Finite Difference discretisation



Meshes and cells

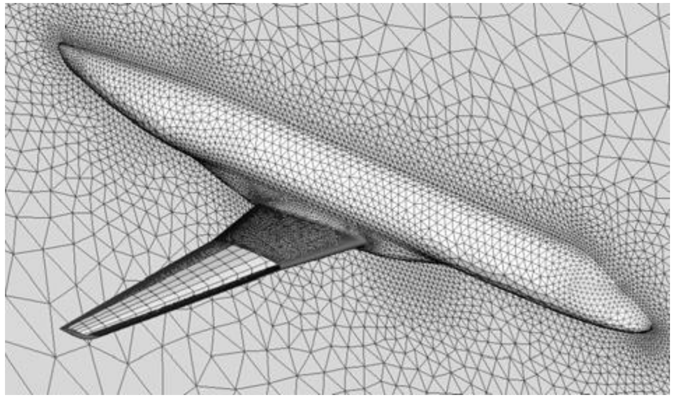
***Definition:** Given a closed, finite, simulation domain, a computational mesh (or grid) is defined as a set of closed, connected volumes (or cells) that completely cover the simulation domain and overlap only along their edges.*



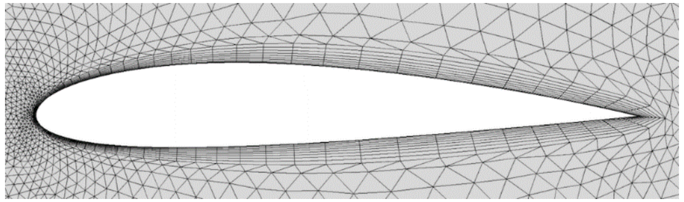
- $\Omega = \bigcup_k V_k$
- $\mathrm{int}(V_j) \cap \mathrm{int}(V_k) = \emptyset, \ \forall j, k$ (int = ‘interior’)
- The process of spatially discretising a simulation domain with a mesh is called mesh generation.
- Various classes of meshes exist, each with their own pros/cons.

UNIVERSITY OF CAMBRIDGE

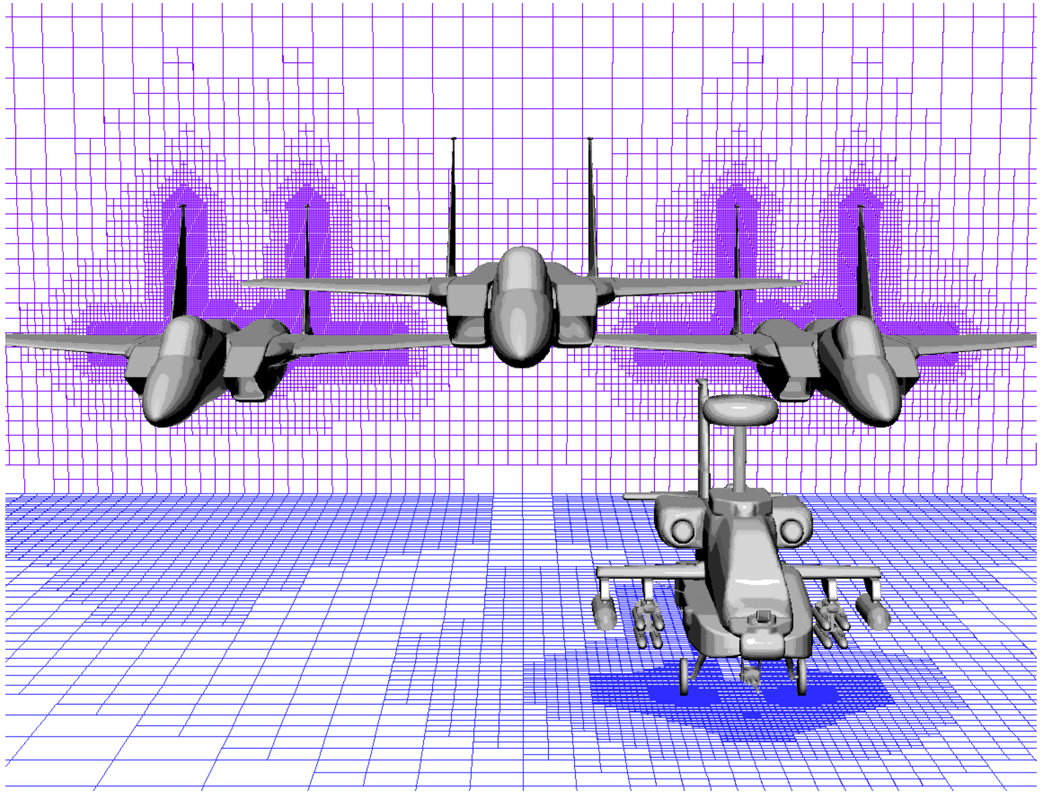
Computational mesh examples



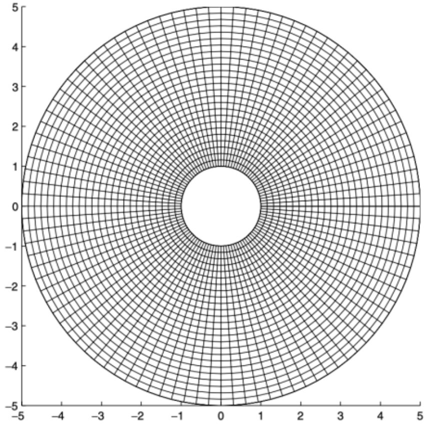
Unstructured [1]



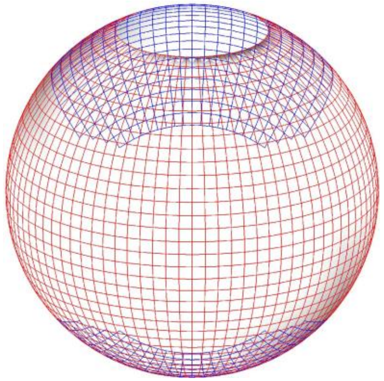
Hybrid unstructured-curvilinear [2]



Block-structured cut-cell [3]



Structured curvilinear [4]



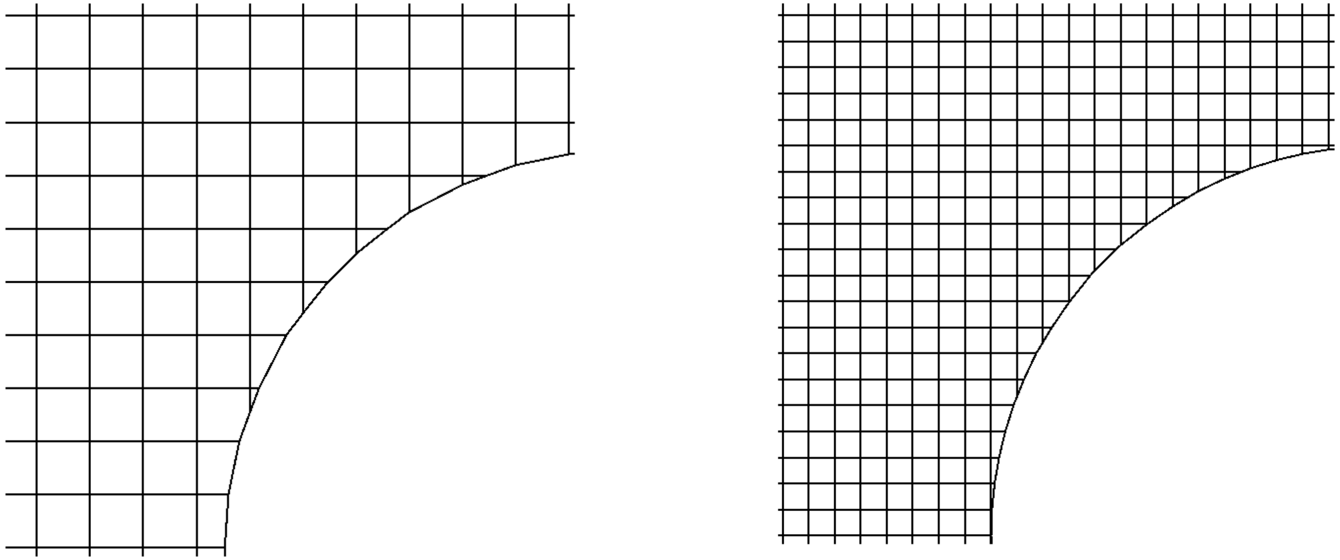
Overlapping curvilinear [5]

UNIVERSITY OF CAMBRIDGE

[1] Obayashi, ECCOMAS CFD, 2006
[2] https://uk.comsol.com/blogs/your-guide-to-meshing-techniques-for-efficient-cfd-modeling/
[3] https://www.nas.nasa.gov/publications/software/docs/cart3d/index.html
[4] LeVeque, R.J., Finite volume methods for hyperbolic problems, CUP, 2012.
[5] Blakely, P.M., et al., General relativistic hydrodynamics on overlapping curvilinear grids. Astronomy & Astrophysics, 2015.

Grid characteristics (1/3)

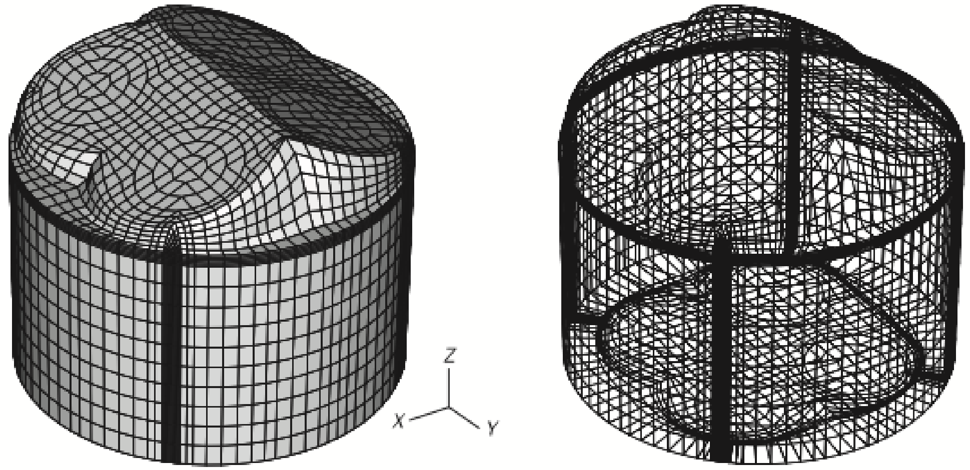
- Solution accuracy is proportional to the density of the grid.



- More cells increase the computational cost of the simulation.

Grid characteristics (2/3)

- Large variations in grid density and/or grid shape are generally associated with reduction of accuracy of numerical solvers.
  - These may lead to stability issues, and phenomena such as spurious reflection or refraction of waves.

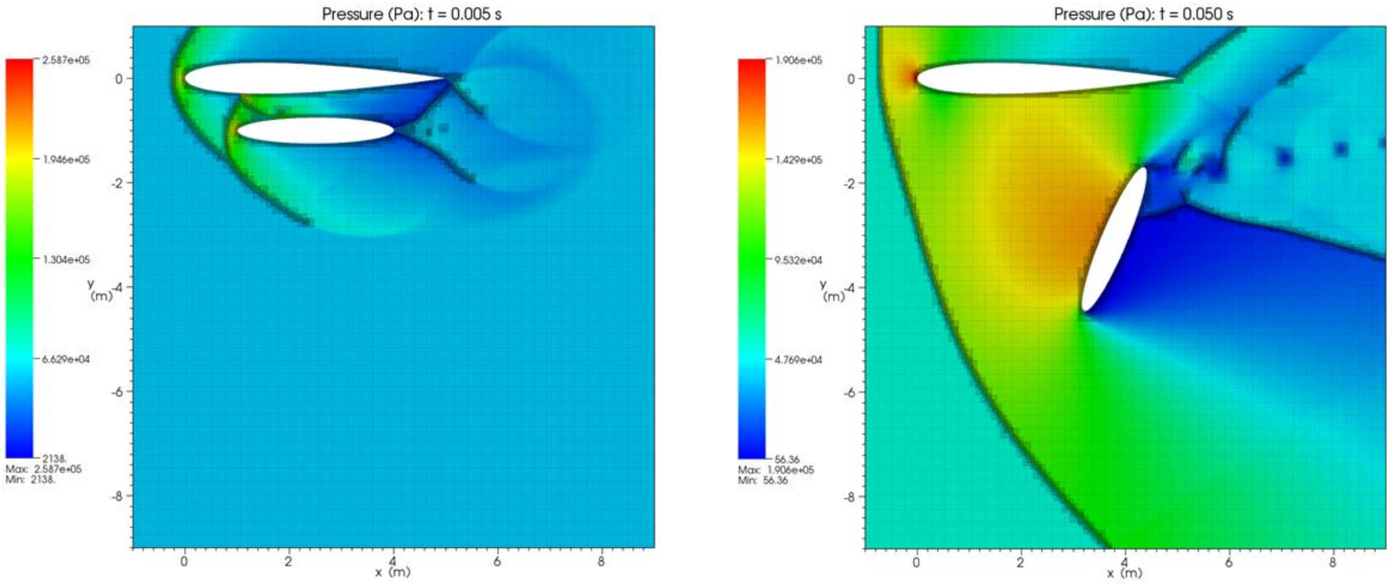


Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson, 2007.

UNIVERSITY OF CAMBRIDGE

Grid characteristics (3/3)

- For simulations involving moving boundaries, grids may have to be regenerated many times over the course of the simulation.



https://www.lsc.phy.cam.ac.uk/sites/www.lsc.phy.cam.ac.uk/files/styles/carousel/public/images/droptank1.png?itok=qn-mIWLB

UNIVERSITY OF CAMBRIDGE

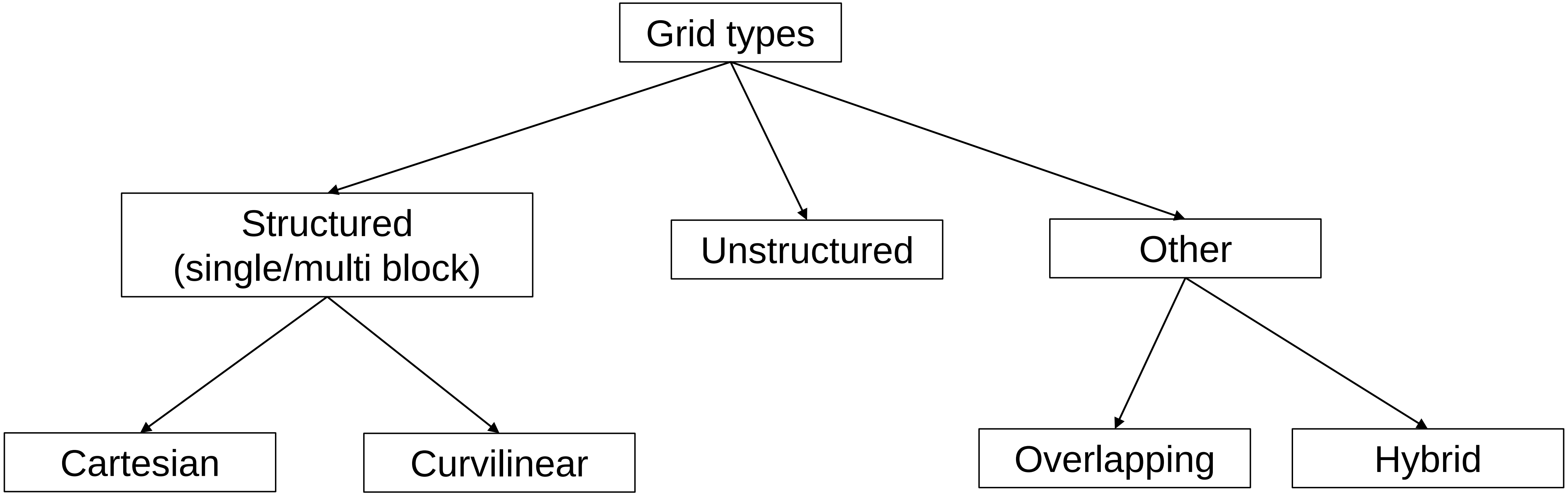
Grid selection

Considerations before selecting a type of grid:

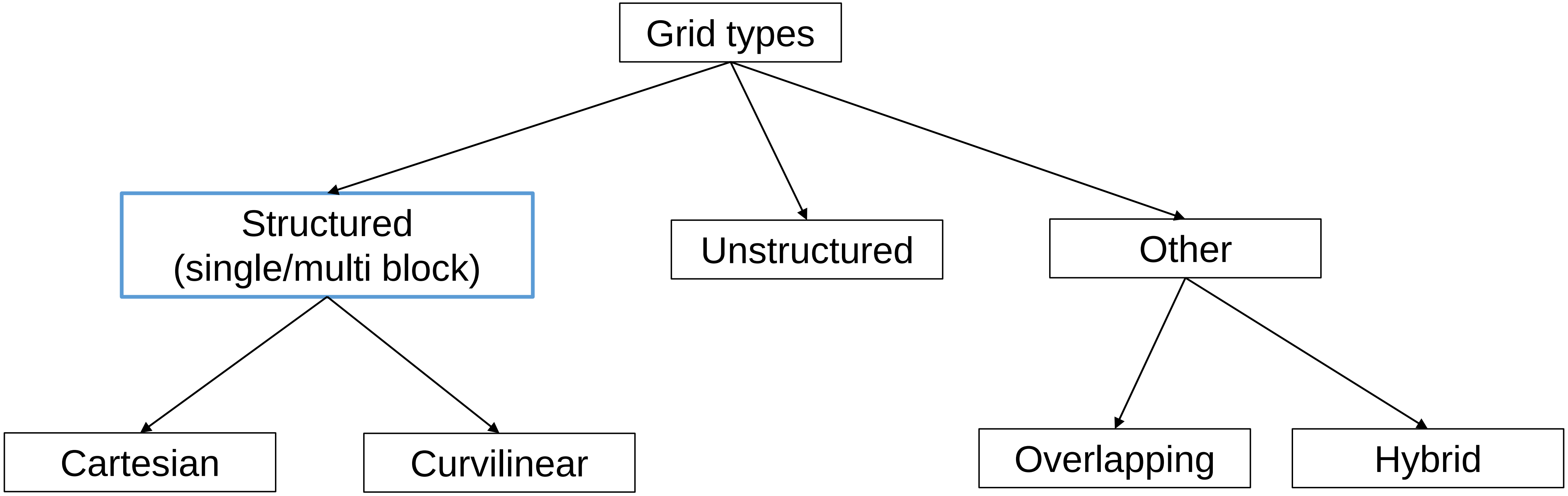
- Viability of generating a good grid (complex geometries, flow features of interest, etc.).
- Implications for the method of solution.
- Associated computational expense.

Grid types

Types of grids



Types of grids

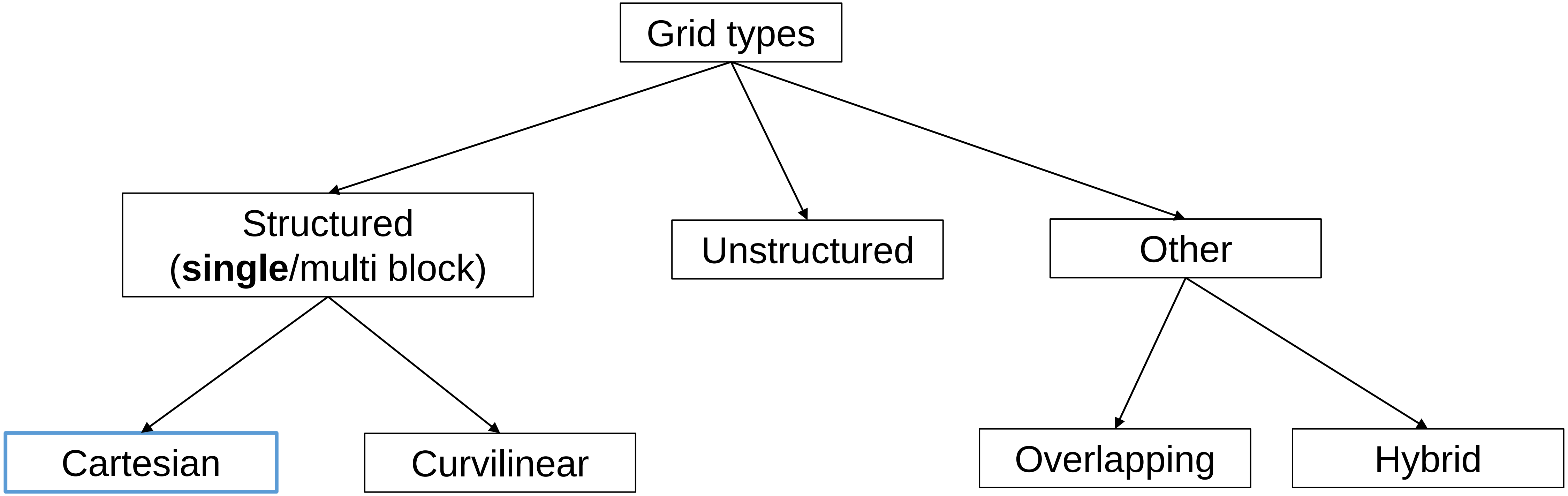


Structured grids

- Cartesian, or logically Cartesian-equivalent grids.
- Grid points can be identified by one, two or three indices (i,j,k), and have two, four or six neighbours depending on the dimensionality.
- The simple connectivity is straightforward and efficient to program (data structures, parallelisation).
- The grid can be a single entity (single-block) or consist of a collection of meshes (multi-block).

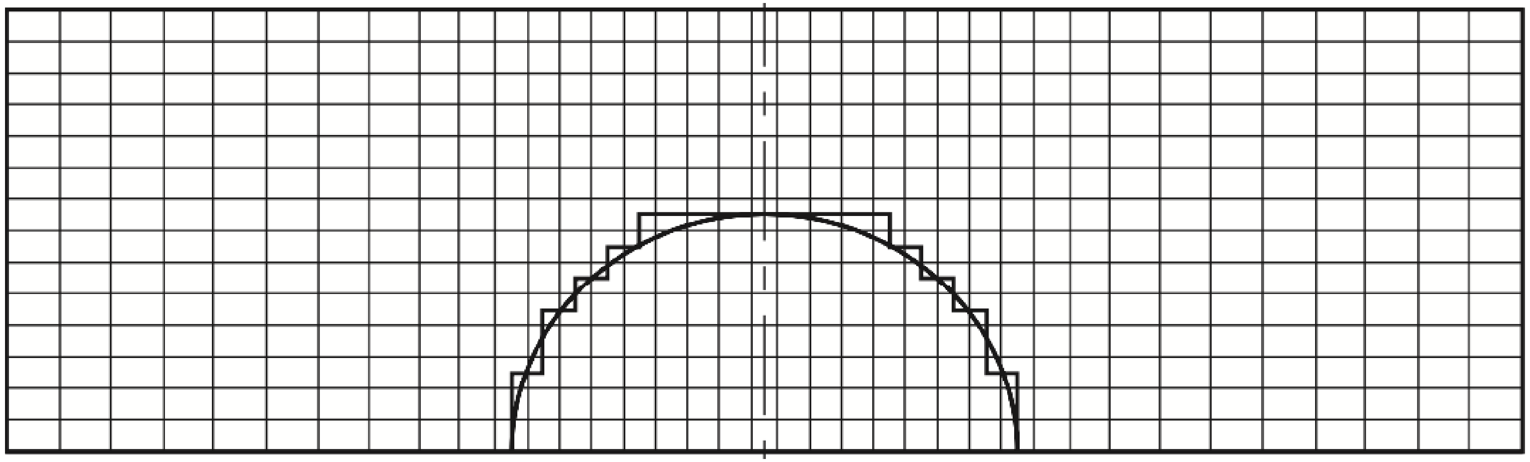
UNIVERSITY OF CAMBRIDGE

Types of grids



UNIVERSITY OF CAMBRIDGE

Structured grids (Step-wise Cartesian)

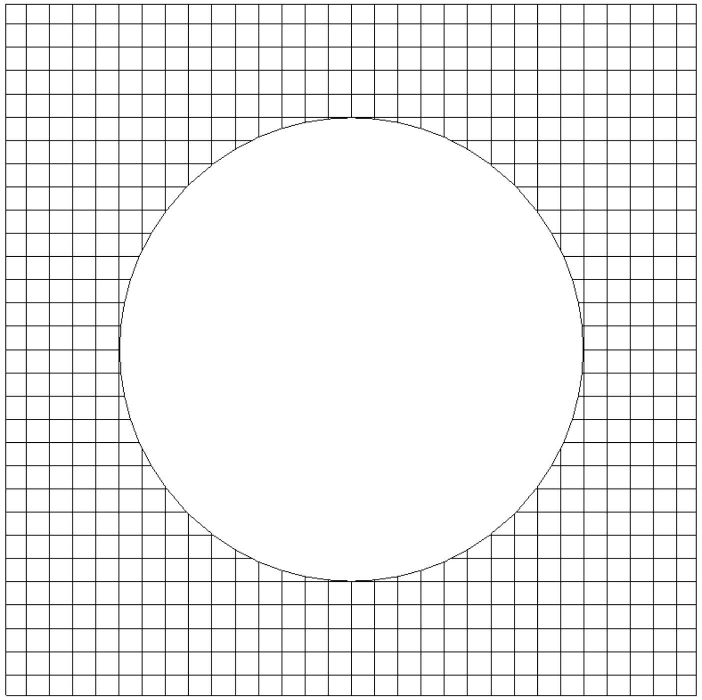


Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson, 2007.

- For non grid-aligned geometries, errors caused through the discontinuous boundary discretisation.
- Features such as boundary layers like to be severely disrupted.

UNIVERSITY OF CAMBRIDGE

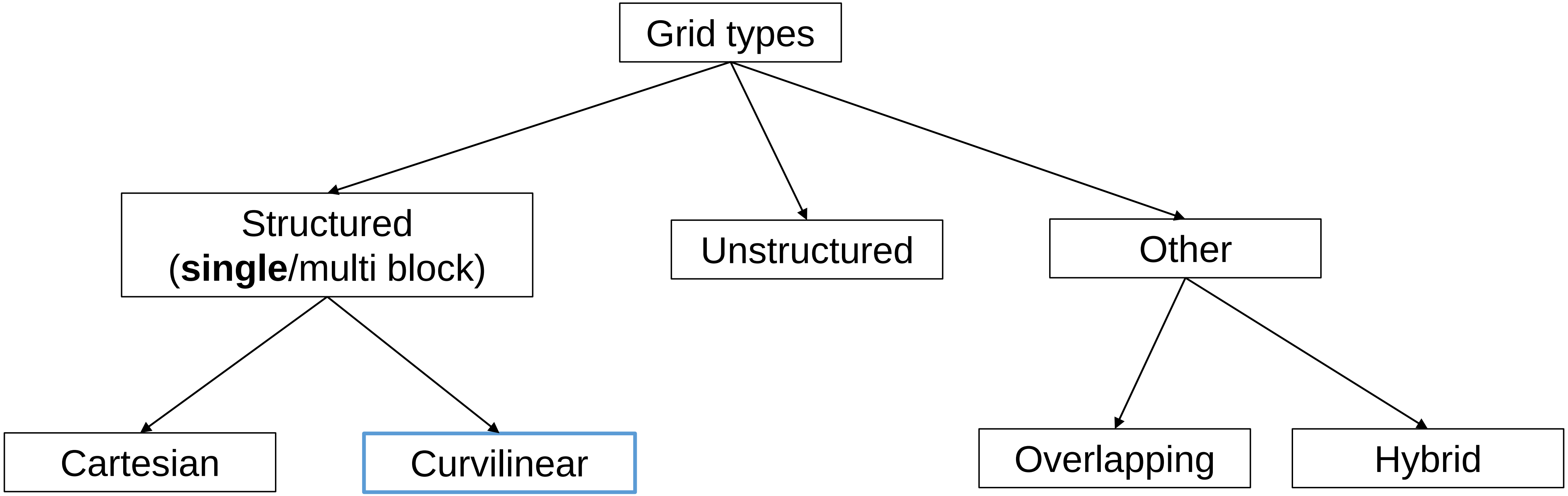
Structured grids (Cartesian cut cell, or embedded boundary)



- Mesh generated by ‘cutting out’ the geometry from a background Cartesian grid.
- Retain the computational convenience of a Cartesian grid, while enabling the discretization of complex domains.
- (Much) more on this later.

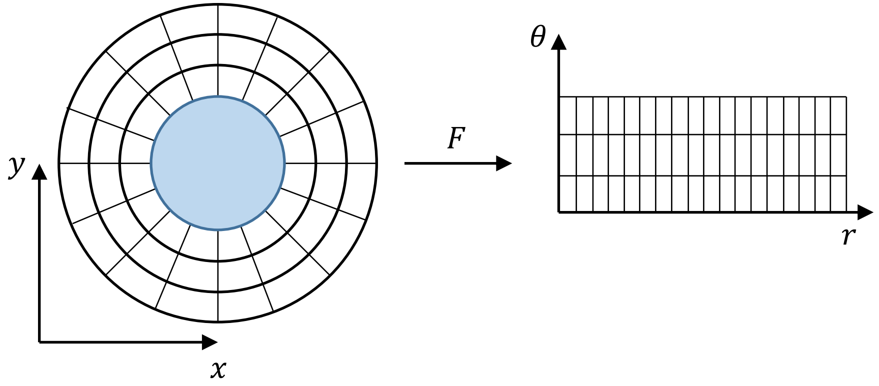
UNIVERSITY OF CAMBRIDGE

Types of grids

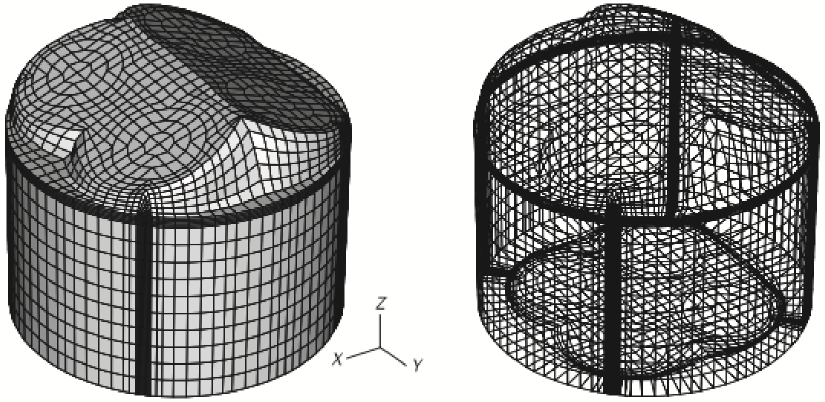


UNIVERSITY OF CAMBRIDGE

Structured grids (Curvilinear, or "body-fitted")



$$(r, \theta) = F(x, y) = \left(\sqrt{x^2 + y^2}, \tan^{-1}\left(\frac{y}{x}\right)\right)$$
$$(x, y) = F^{-1}(r, \theta) = (r\cos\theta, r\sin\theta)$$



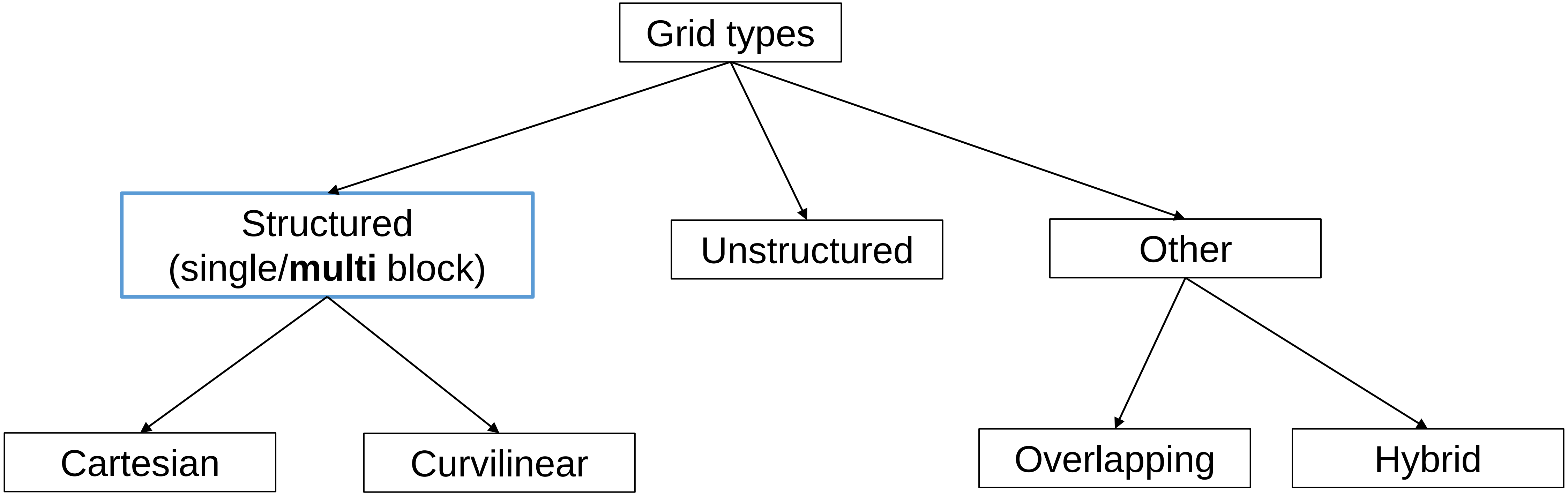
Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson, 2007.

- Physical domain mapped to a rectangular computational domain using an invertible coordinate transformation.
- The equations are discretised and solved in the computational space.

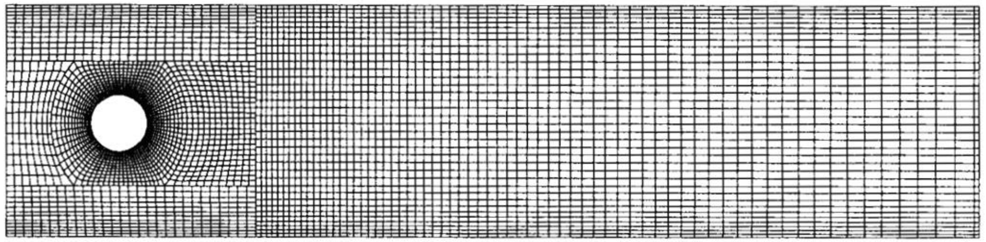
- May be impossible to find a suitable mapping for a complex geometry.
- Mesh skewness and unnecessarily high grid resolutions where mapping is difficult (e.g. near sharp corners).

UNIVERSITY OF CAMBRIDGE

Types of grids

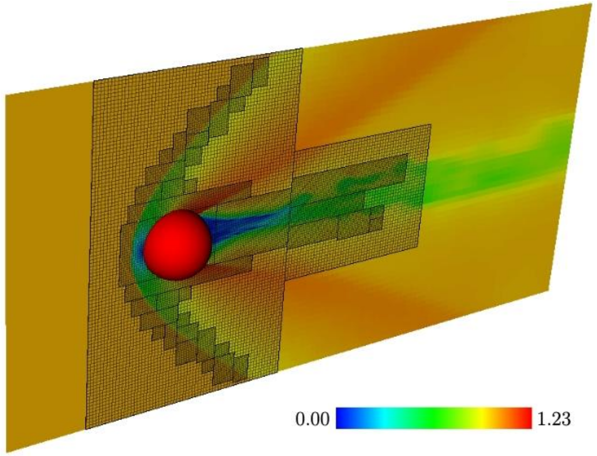


Structured grids (multi-block)



Multiple curvilinear blocks for the simulation of flow over a cylinder in a narrow duct [1].

Block-structured Cartesian cut cell mesh for the simulation of flow over a sphere [2].

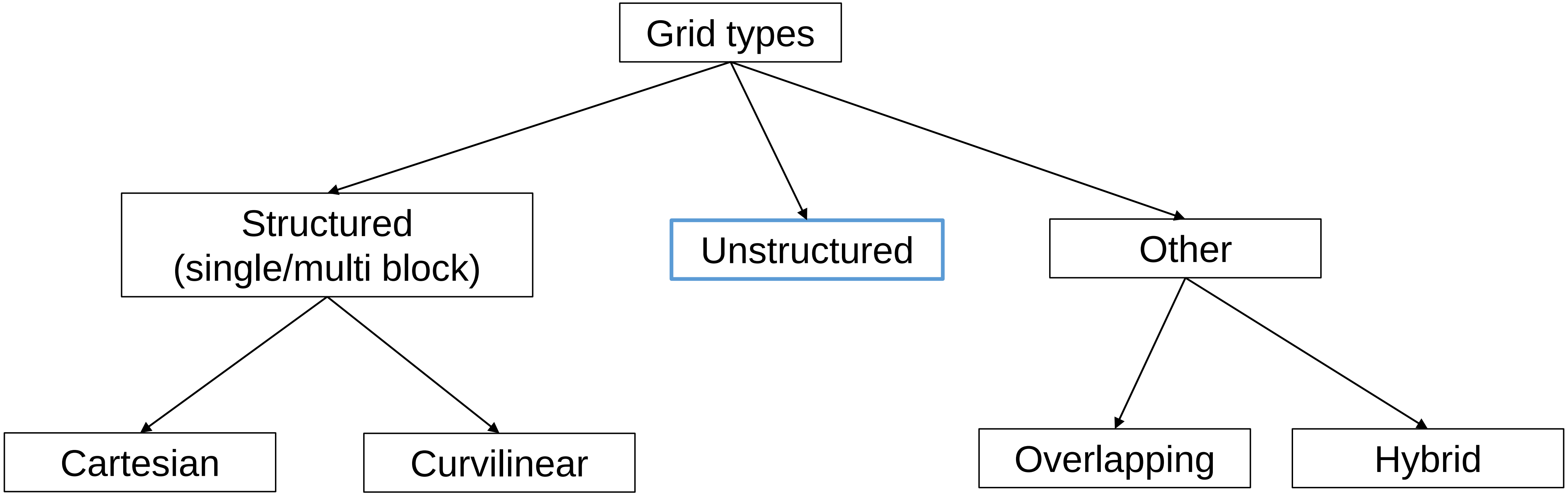


- Use a combination of several single block grids that best fit different regions of the domain.
- The cells at block interfaces may or may not be matched, and must be properly treated in a fully conservative manner.
- Block-structured AMR techniques will be discussed in another lecture.

UNIVERSITY OF CAMBRIDGE

[1] Ferziger, J.H. and Peric, M., Computational Methods for Fluid Dynamics, Springer, 2002.
[2] Gokhale et al., A dimensionally split Cartesian cut cell method for the compressible Navier–Stokes equations, J. Comput. Phys, 2018.

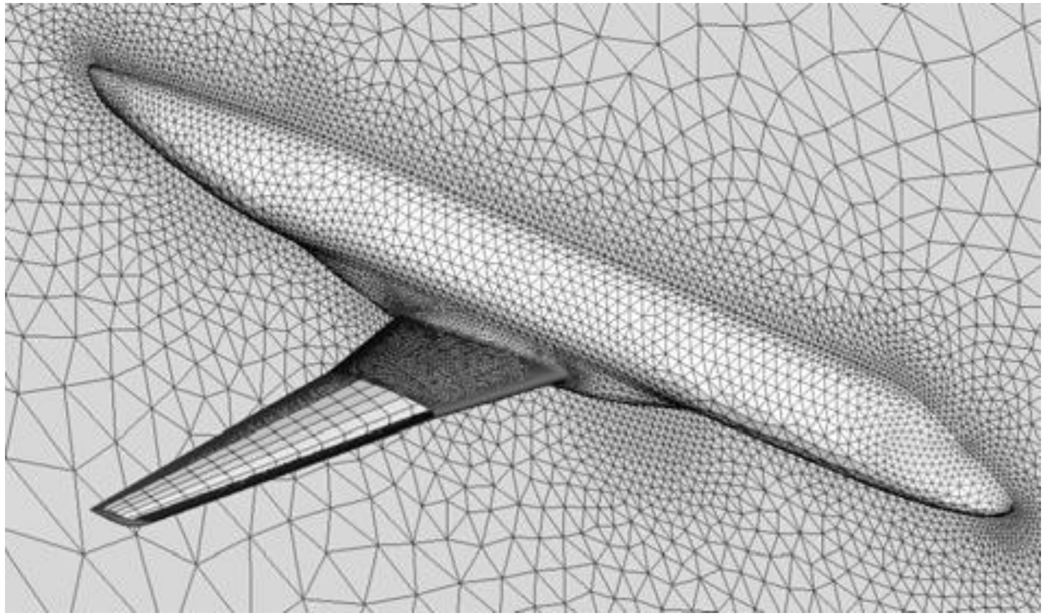
Types of grids



UNIVERSITY OF CAMBRIDGE

Unstructured meshes

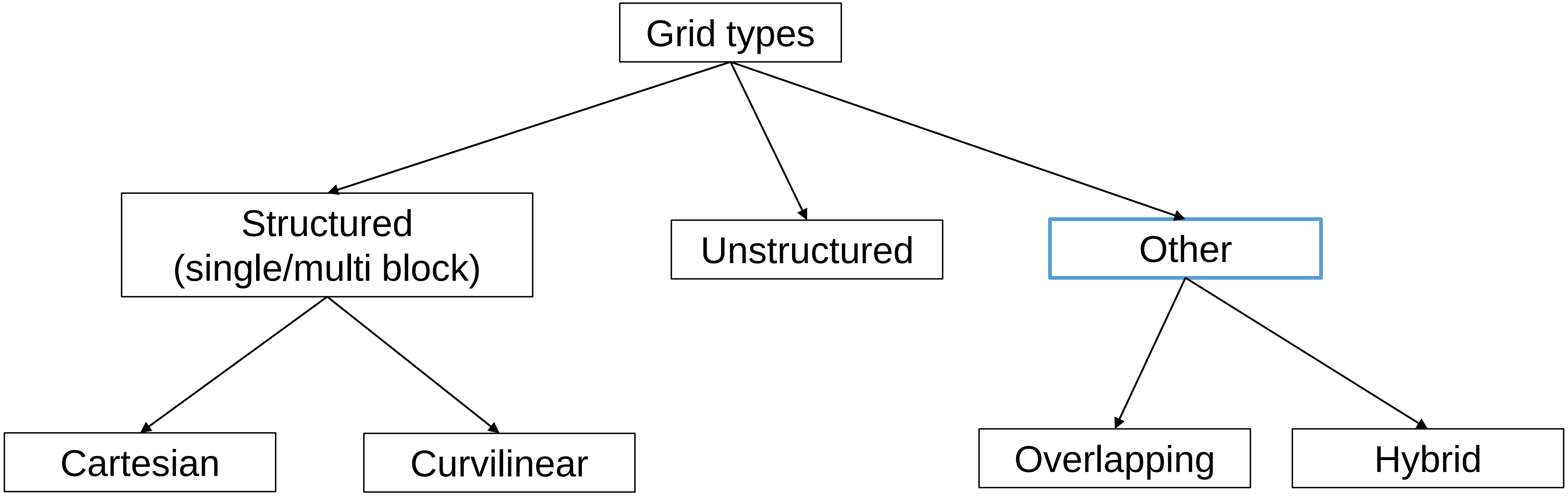
Definition: *A grid is structured iff it is topologically equivalent to a Cartesian grid. Otherwise, it is referred to as an unstructured grid.*



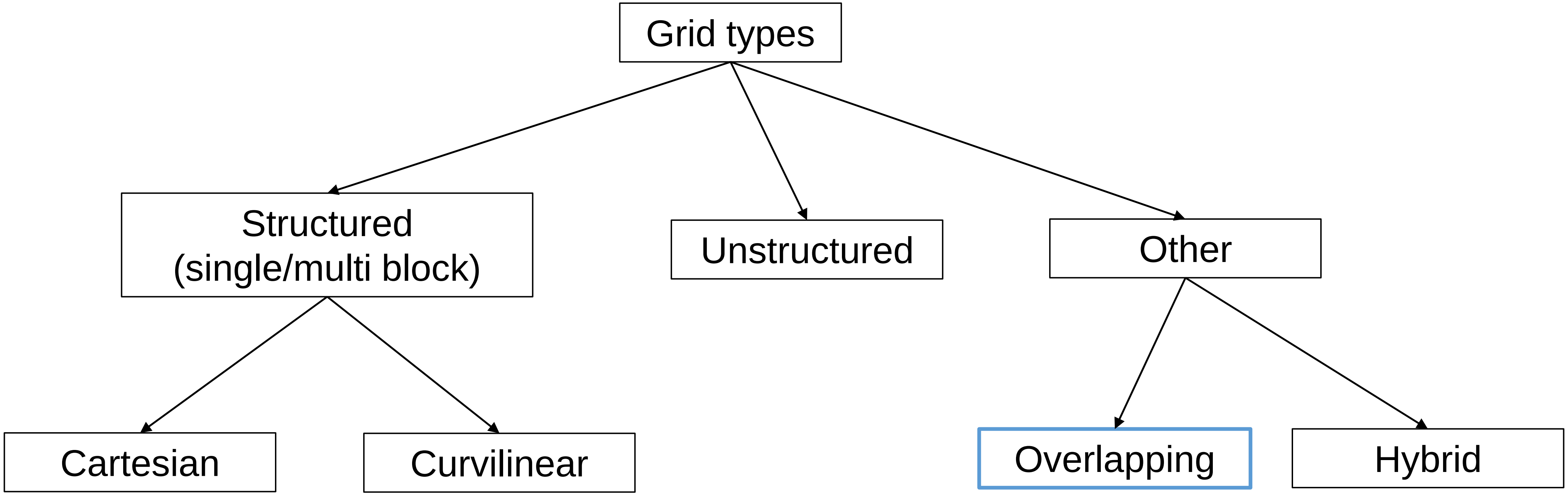
Obayashi, ECCOMAS CFD, 2006

- One of the most widely used techniques to discretise complex domains (discretization achieved with polyhedral cells).
- Important considerations: data structures, mesh connectivity, mesh quality metrics.
- Covered in another course.

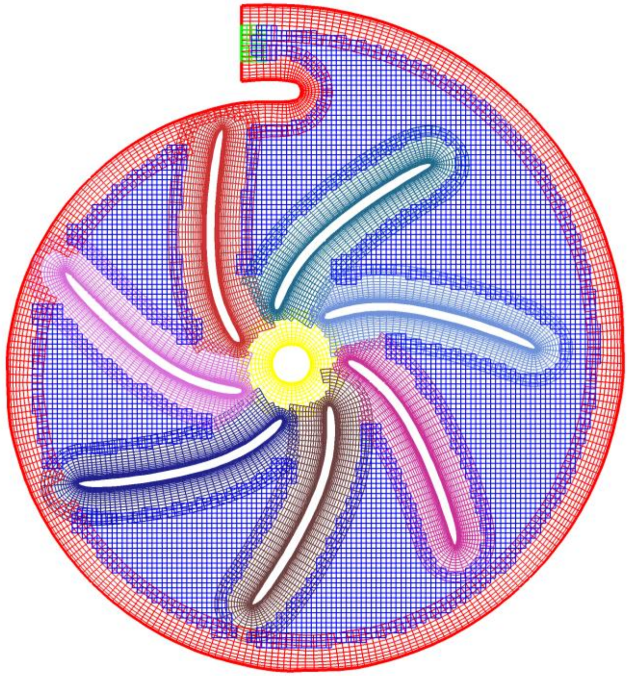
Types of grids



Types of grids



Other grids (overlapping)

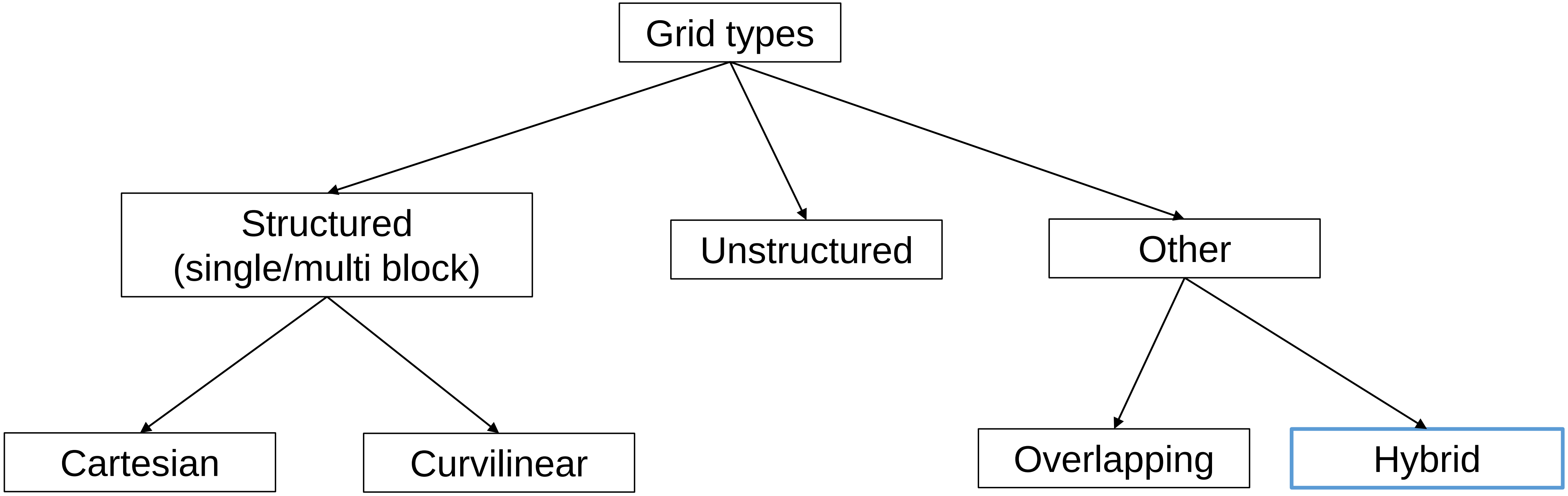


https://overtureframework.org/figures/pump2dGrid.gif

- Similar to structured multi-block, but now, neighbouring blocks overlap.
- Particularly useful with moving boundaries, since only meshes local to the geometry need to be modified.
- Interpolation based on assigned grid priorities used to calculate the solution in the overlap regions. Ideally, the interpolation should be conservative (achieving this is not straightforward).
- Other titles: overset, overlaid, Chimera grids.

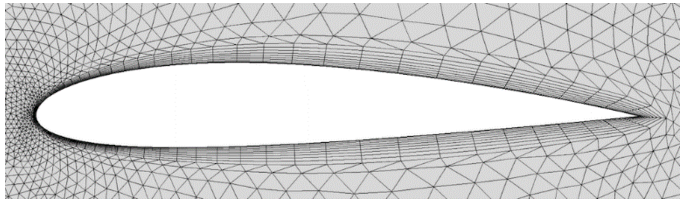
UNIVERSITY OF CAMBRIDGE

Types of grids

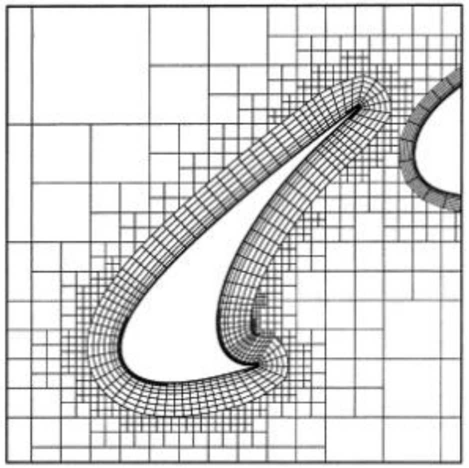


UNIVERSITY OF CAMBRIDGE

Other grids (hybrid)



Hybrid unstructured-curvilinear [1]



Hybrid Cartesian-curvilinear [2]

- Composite grid created by combining structured and unstructured grids in different parts of the domain.

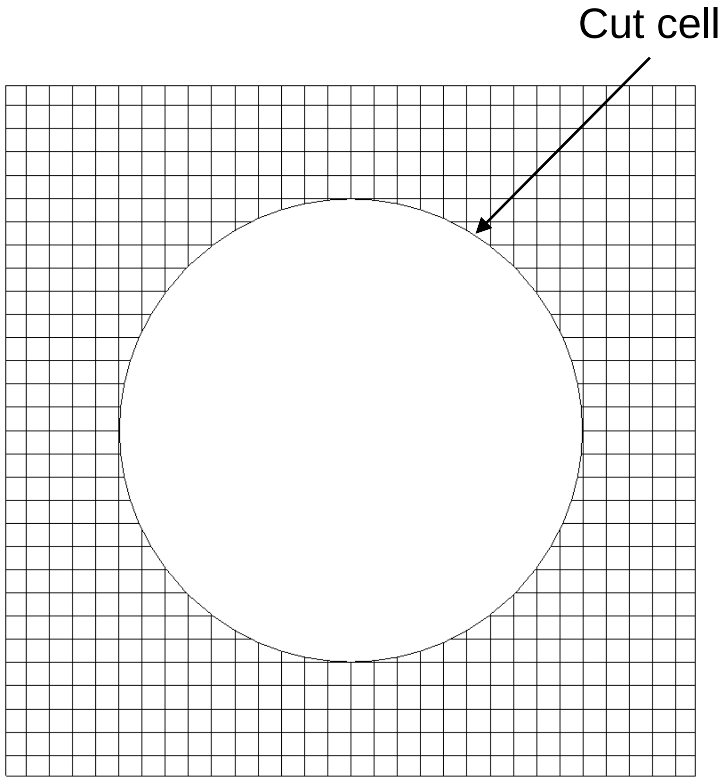
- The primary motivation for this approach is to accurately resolve the boundary layer in a viscous flow without the difficulty of discretising the entire domain with the body-fitted mesh.

UNIVERSITY OF CAMBRIDGE

[1] https://uk.comsol.com/blogs/your-guide-to-meshing-techniques-for-efficient-cfd-modeling/
[2] Zang, A Quadtree-based adaptive Cartesian/Quad grid flow solver for Navier-Stokes equations, Computers & Fluids, 1998.

Cartesian cut cell methods (fundamentals)

Cut cell meshes and methods



- (Recall) Mesh generated by ‘cutting out’ the geometry from a background Cartesian grid.

- As we shall see, the possible existence of arbitrarily small cut cells makes calculating the solution on a cut cell mesh is tricky. Any method used to do this is called a ‘cut cell method’.

- Cut cell methods are classified as ‘sharp interface methods’ (sharp and diffuse interface methods will be covered in other lectures).

UNIVERSITY OF CAMBRIDGE

Cartesian “Immersed Boundary Methods”

{Sharp Int., Diffuse Int.}
x
{FV, FD}
x
{Cons., Non cons.}
x
{Hyp., Non-Hyp.}

IB methods

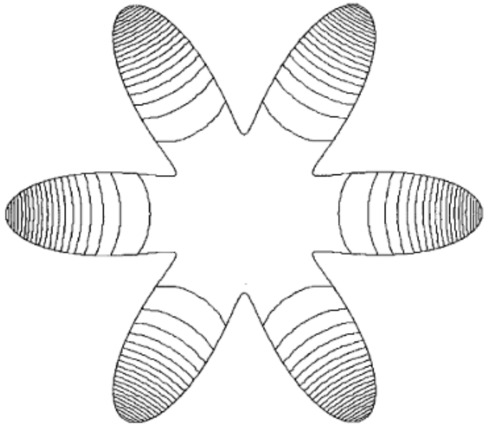
- At least 16 types of IB methods.
- Cut cell method: (Sharp Int., FV, Cons., Hyp./Non-Hyp.).
- However, the vocabulary and ideas are very transferable.

UNIVERSITY OF CAMBRIDGE

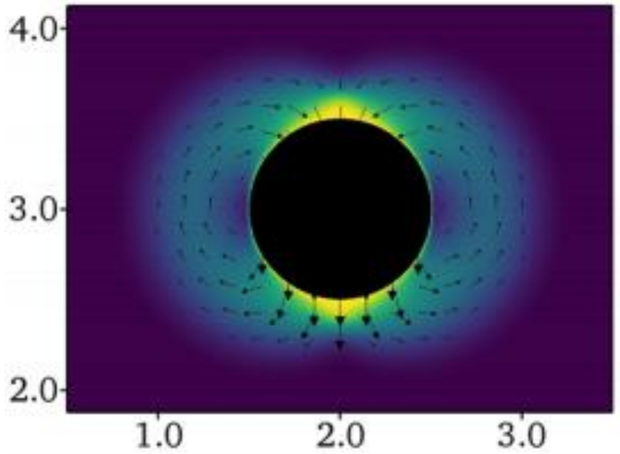
Cut cell methods for hyperbolic and non-hyperbolic systems

- For implicitly integrated parabolic parts, or purely elliptic parts, of a system, we have to ensure that the equation discretisation at the cut cells is such that the matrix is well-conditioned. This allows the system to be solved efficiently using multigrid, for example (see Johansen and Colella [1] and Graves et al. [2]).
  - The solution of linear systems of equations is covered in another course.

Solution of Poisson's equation (elliptic) in a complex domain [1]



Cylinder moving through a Bingham fluid. Part of this incompressible non-Newtonian system is solved elliptically [3]



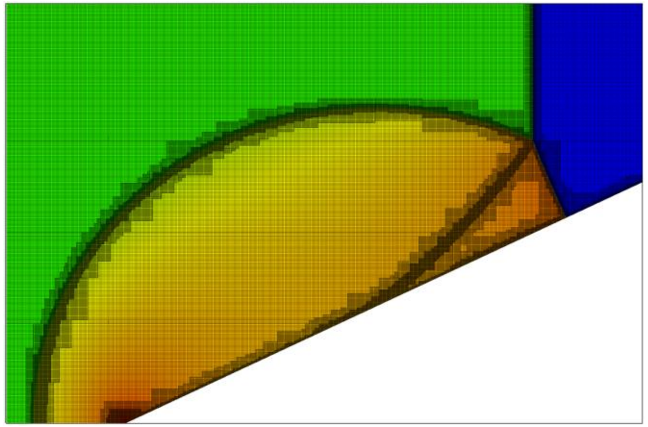
UNIVERSITY OF CAMBRIDGE

[1] Johansen, H. and Colella, P., A Cartesian grid embedded boundary method for Poisson's equation on irregular domains, J. Comput. Phys, 1998.
[2] Graves, D. et al., A cartesian grid embedded boundary method for the compressible Navier–Stokes equations, Comm. App. Math. and Comput. Sc., 2013.
[3] Sverdrup, K. et al., An embedded boundary approach for efficient simulations of viscoplastic fluids in three dimensions. Phys. Fluids, 2019.

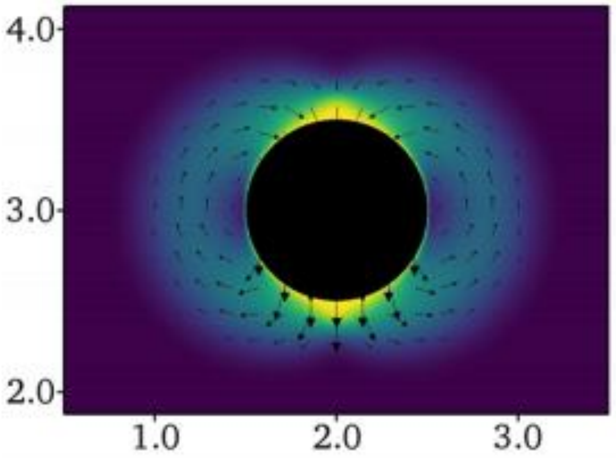
Cut cell methods for hyperbolic and non-hyperbolic systems

- In these lectures, we will focus on techniques that address systems being integrated explicitly in time (hyperbolic or explicitly integrated parabolic parts of systems).
- If a cut cell method wasn't used, the maximum stable Δt would be proportional to the size of the smallest cut cell ('small cell problem').

M=1.7 shock diffraction over a wedge (hyperbolic system)



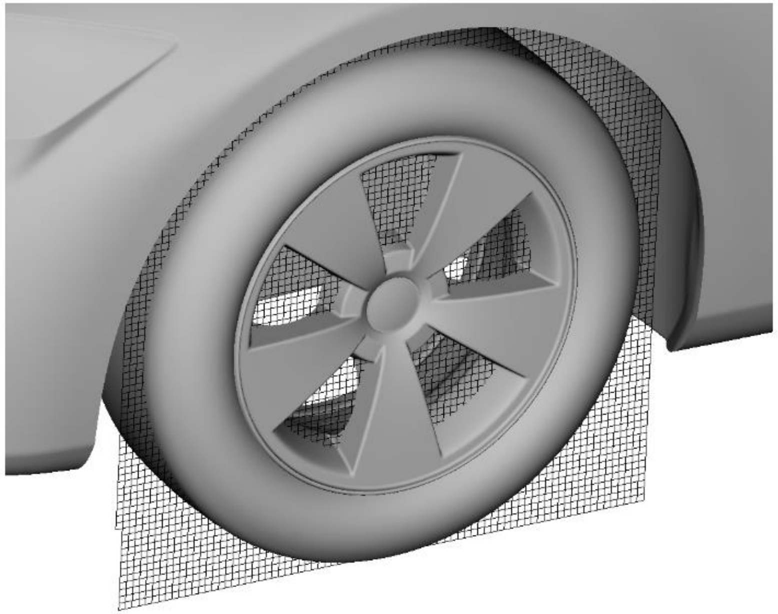
Cylinder moving through a Bingham fluid. Part of this incompressible non-Newtonian system is solved in a hyperbolic fashion [1]



UNIVERSITY OF CAMBRIDGE

[1] Sverdrup, K. et al., An embedded boundary approach for efficient simulations of viscoplastic fluids in three dimensions. Phys. Fluids, 2019.

Advantages of cut cell methods

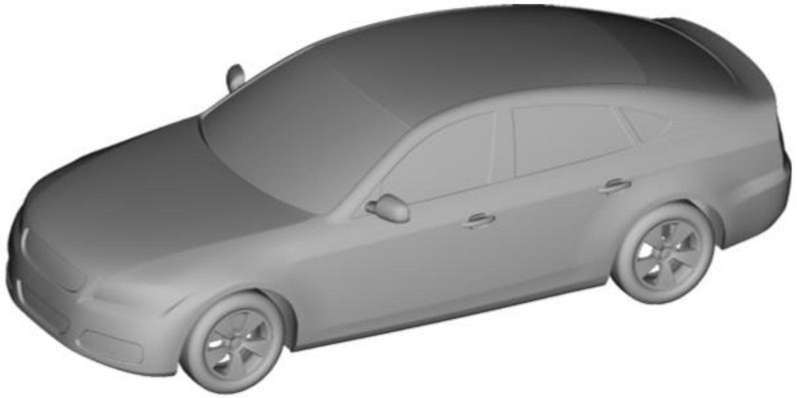


- Facilitate automatic mesh generation, i.e., the rapid creation of accurate meshes with no user intervention.
- Retain the computational conveniences of structured Cartesian grids while allowing the discretisation of complex domains in a non-stepwise fashion.
- The stable Δt is still determined by the size of an 'uncut' cell, and is not constrained by the smallest cell size.
- Grid re-generation (computing new cuts) for moving boundary problems can be done locally and efficiently [1] (we will not look into this aspect).

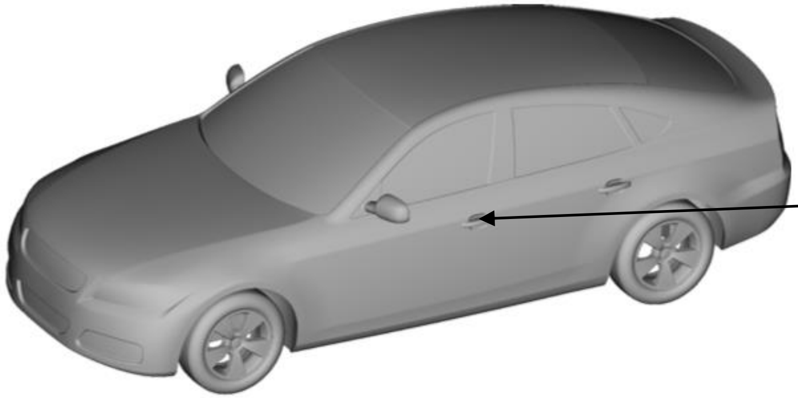
UNIVERSITY OF CAMBRIDGE

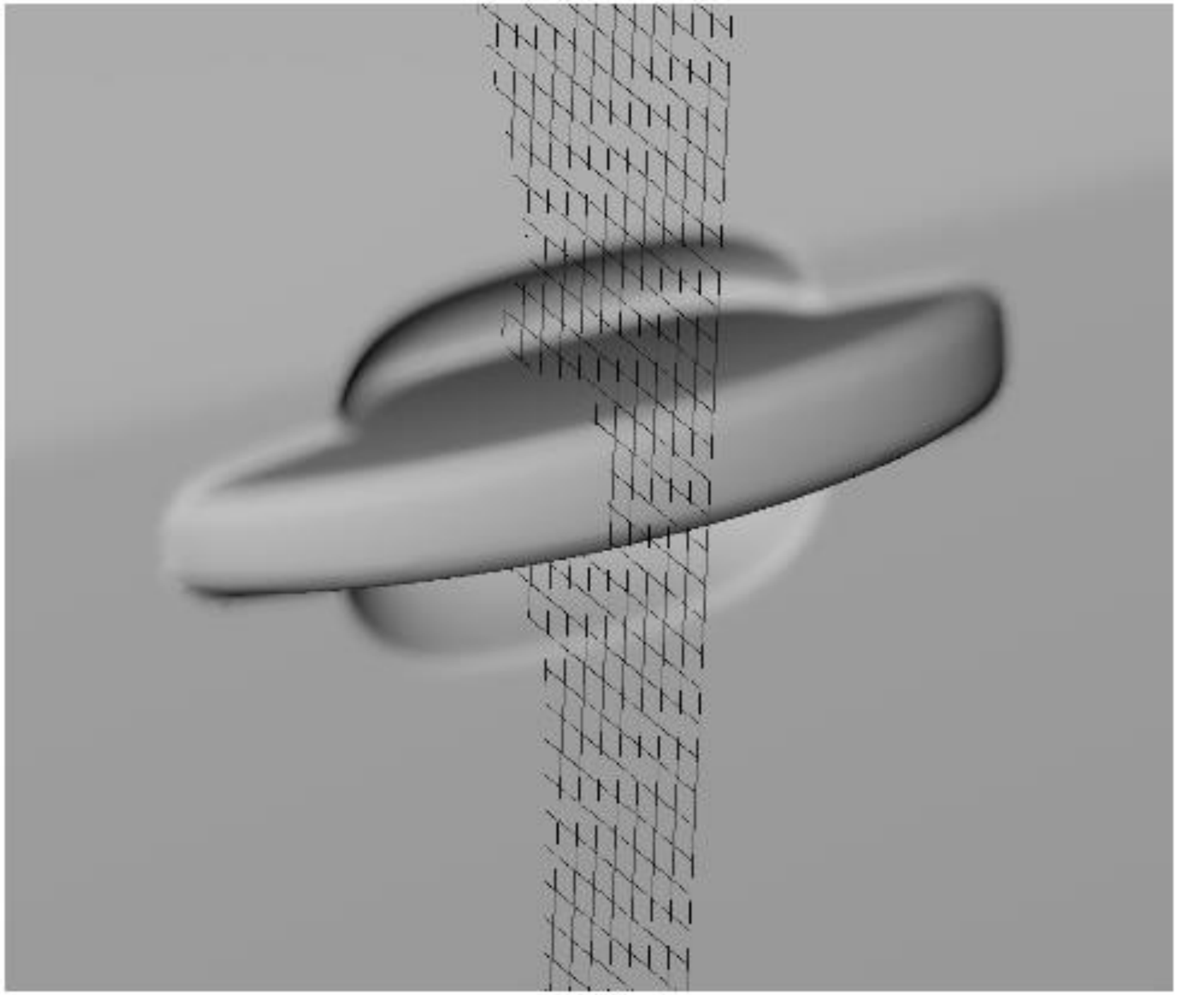
[1] Bennett et al., A moving boundary flux stabilization method for Cartesian cut-cell grids using directional operator splitting, J. Comput. Phys, 2018.

Cut cell mesh examples (1/2) - DrivAer



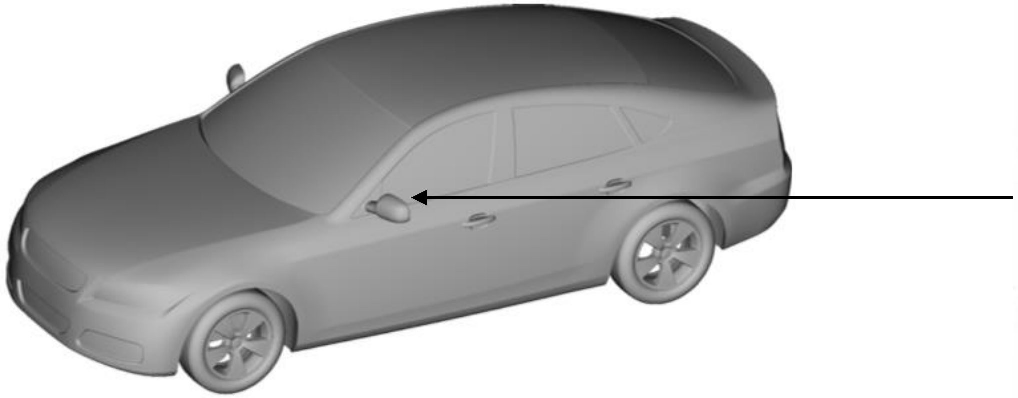
Cut cell mesh examples (1/2) - DrivAer

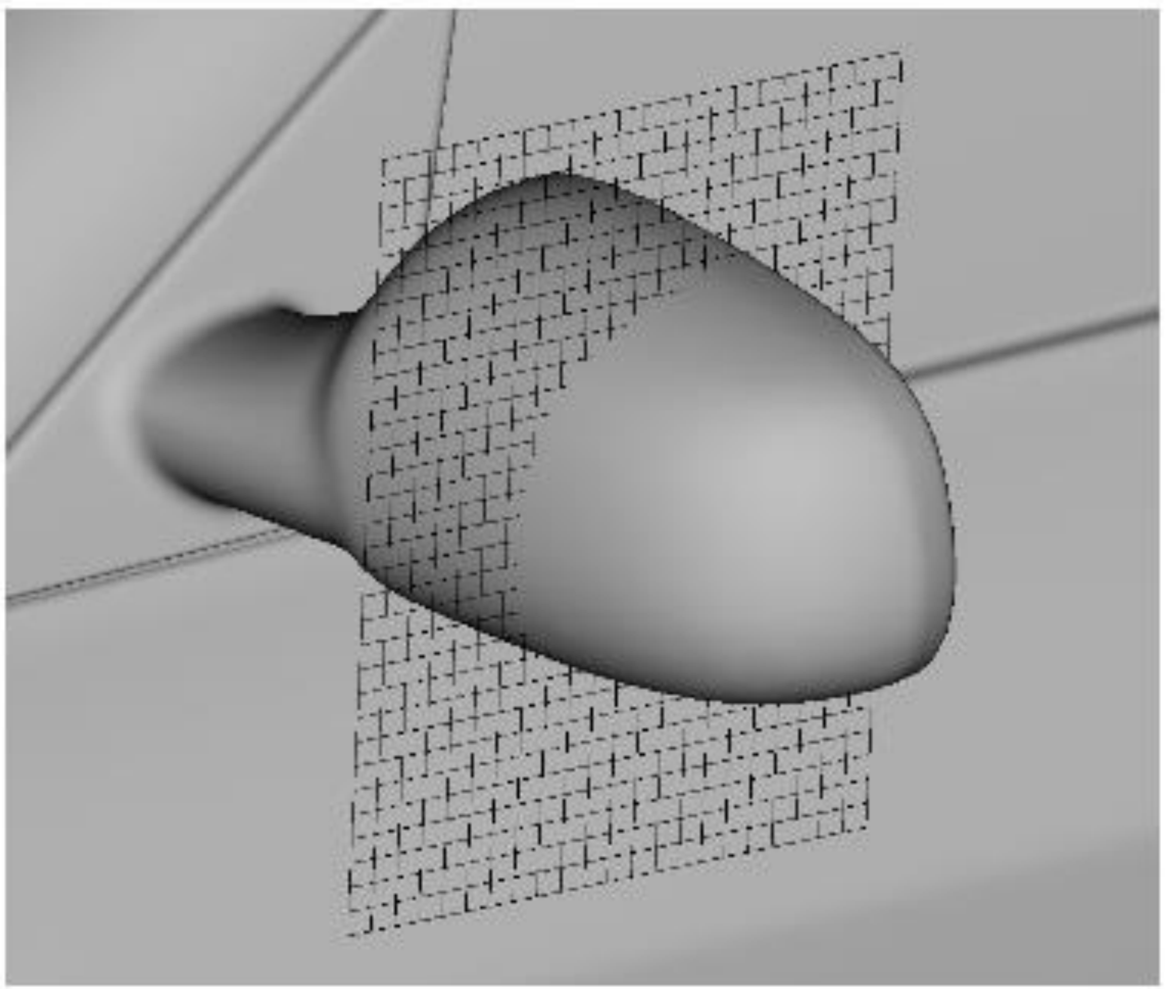




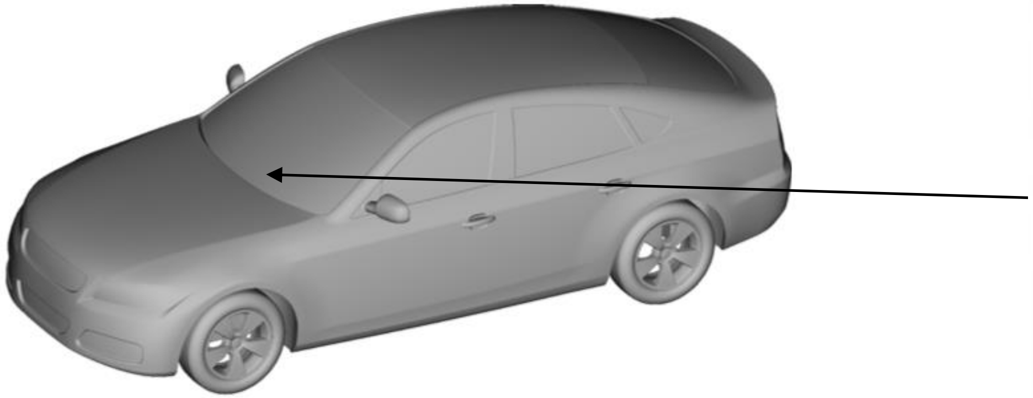
University of Cambridge

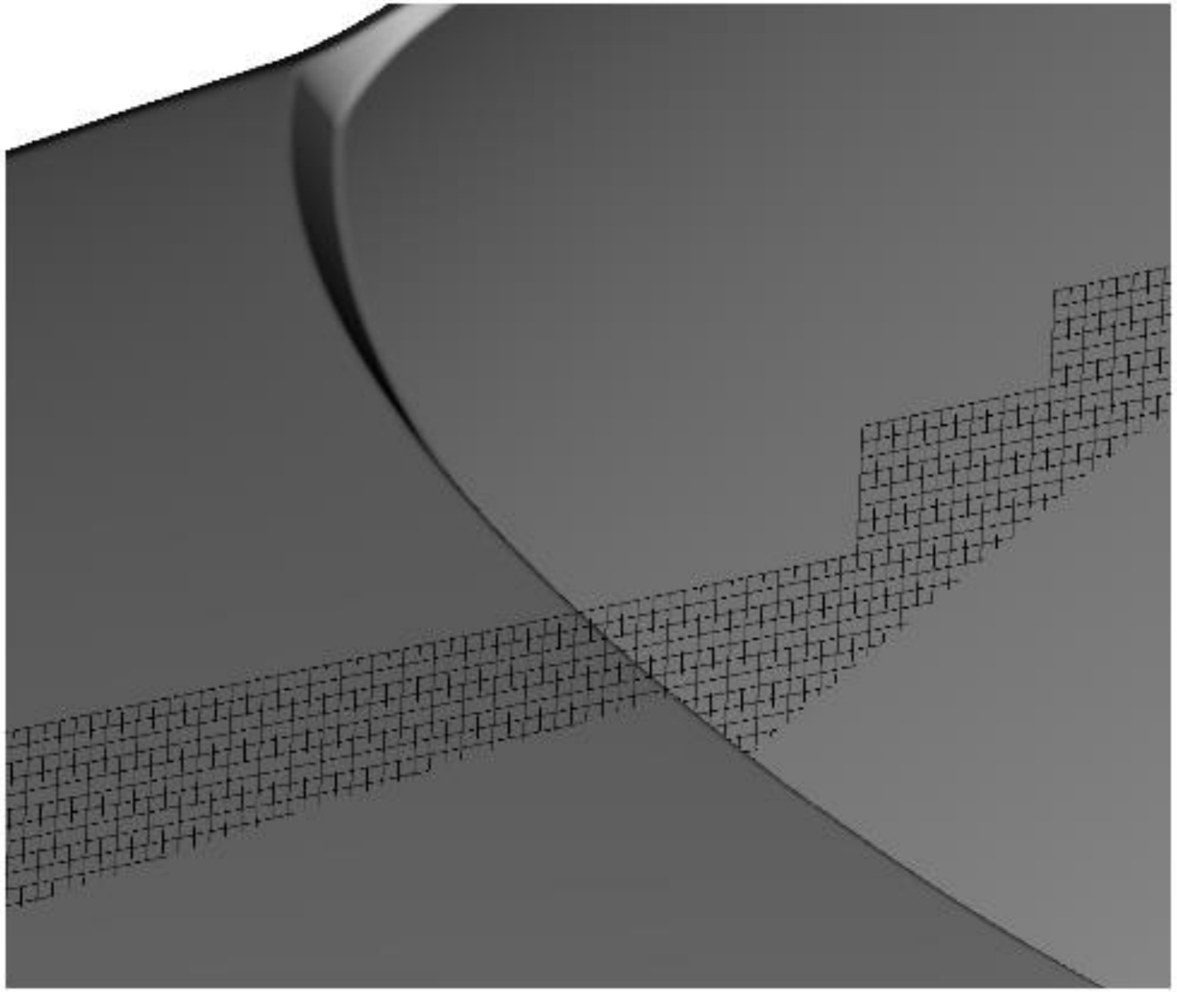
Cut cell mesh examples (1/2) - DrivAer



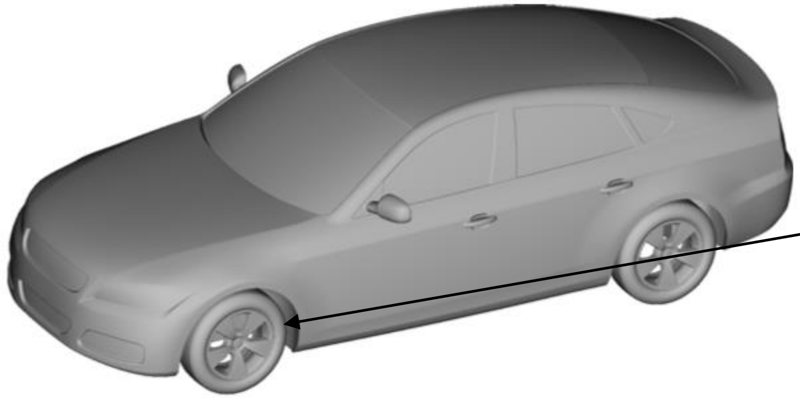


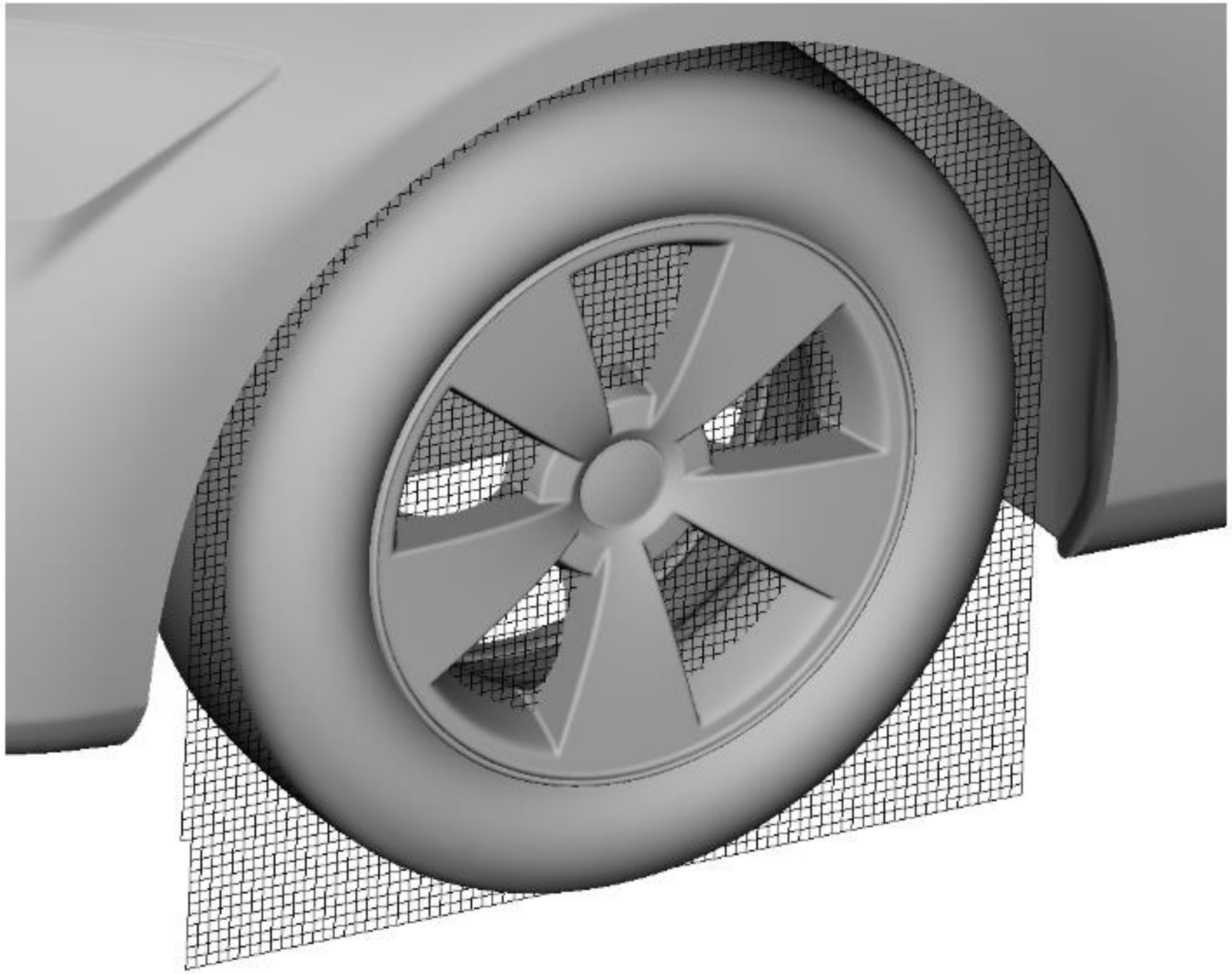
Cut cell mesh examples (1/2) - DrivAer





Cut cell mesh examples (1/2) - DrivAer





UNIVERSITY OF CAMBRIDGE

Cut cell mesh examples (1/2) - DrivAer

Mesh with ~20 million cells created in <30 seconds with no user input.

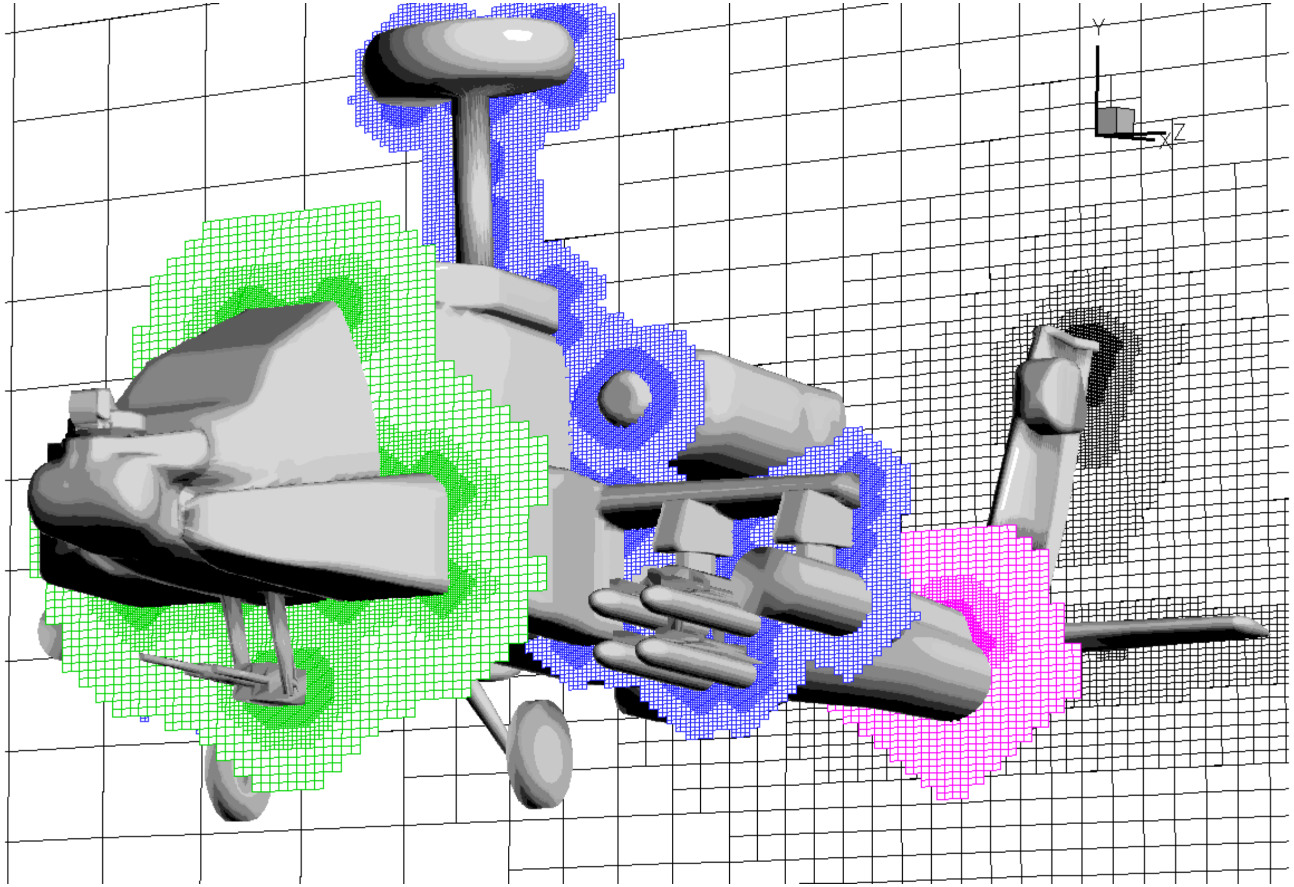
Tesla K20c GPU

Cut cell mesh examples (2/2) – Cart3D

- Used in every new aeronautic design project at NASA Ames.
  (Berger, Handbook of Numerical Analysis, 2017)

- Main authors: Marsha Berger and Michael Aftosmis.

- Still a restricted technology!

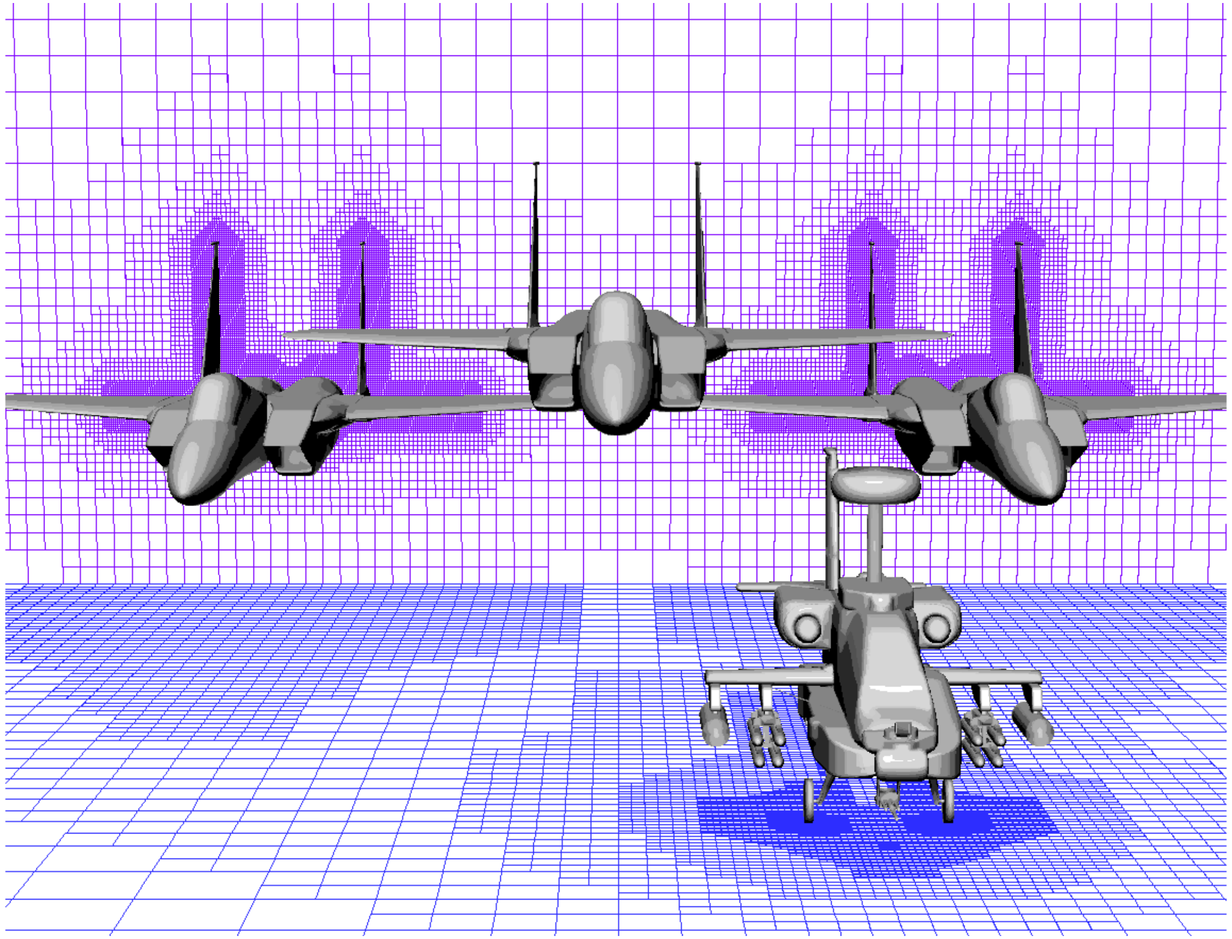


Apache, 5.81M cells

https://www.nas.nasa.gov/publications/software/docs/cart3d/index.html

UNIVERSITY OF CAMBRIDGE

Cut cell mesh examples (2/2) – Cart3D



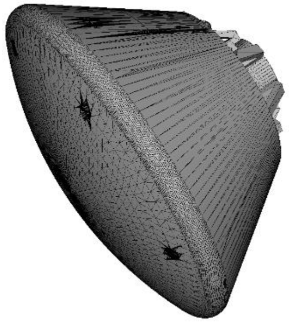
https://www.nas.nasa.gov/publications/software/docs/cart3d/index.html

UNIVERSITY OF CAMBRIDGE

Cut cells automatic mesh generation

STL file (typically)
Common CAD format that contains the coordinates of the vertices and normals of the surface triangulation.

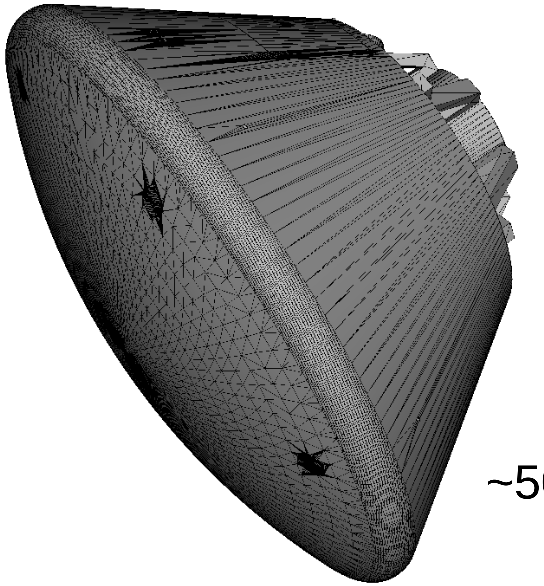
→ Cut cell mesh generator → Cut cell geometric parameters (volume, normal, etc.)



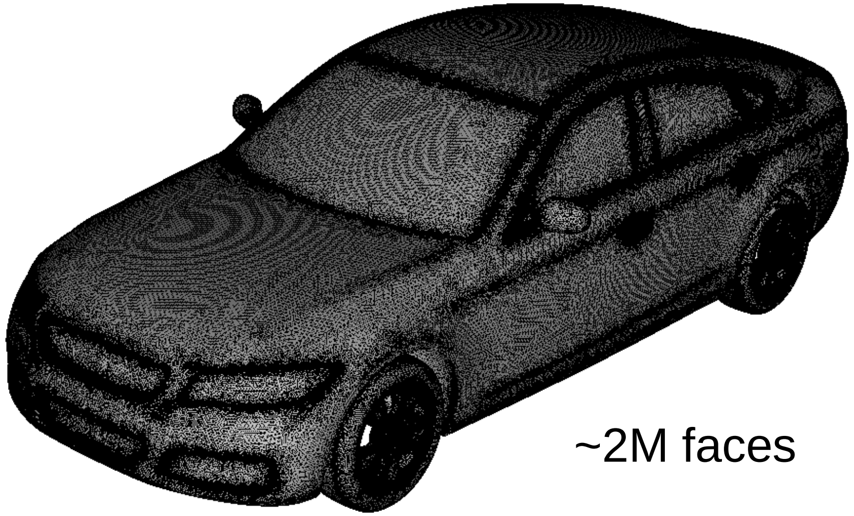
UNIVERSITY OF CAMBRIDGE

Cut cells automatic mesh generation (STL files)

- The STL should be “watertight”, i.e., each edge should be part of exactly 2 triangles, and there should be no self-intersections.
- Unfortunately, most complicated STL files aren’t.



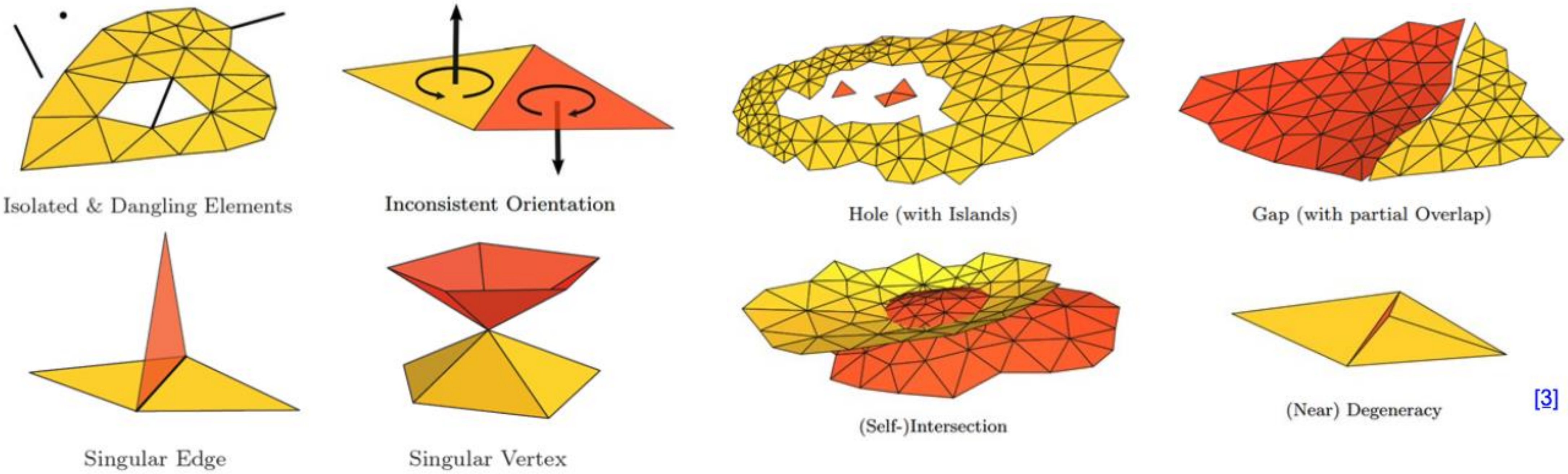
~50k faces



~2M faces

UNIVERSITY OF CAMBRIDGE

Cut cells automatic mesh generation (STL file issues)



M. Attene, 2010, https://slideplayer.com/slide/7743202/

- CAD repair tools available (e.g. Autodesk Netfabb).

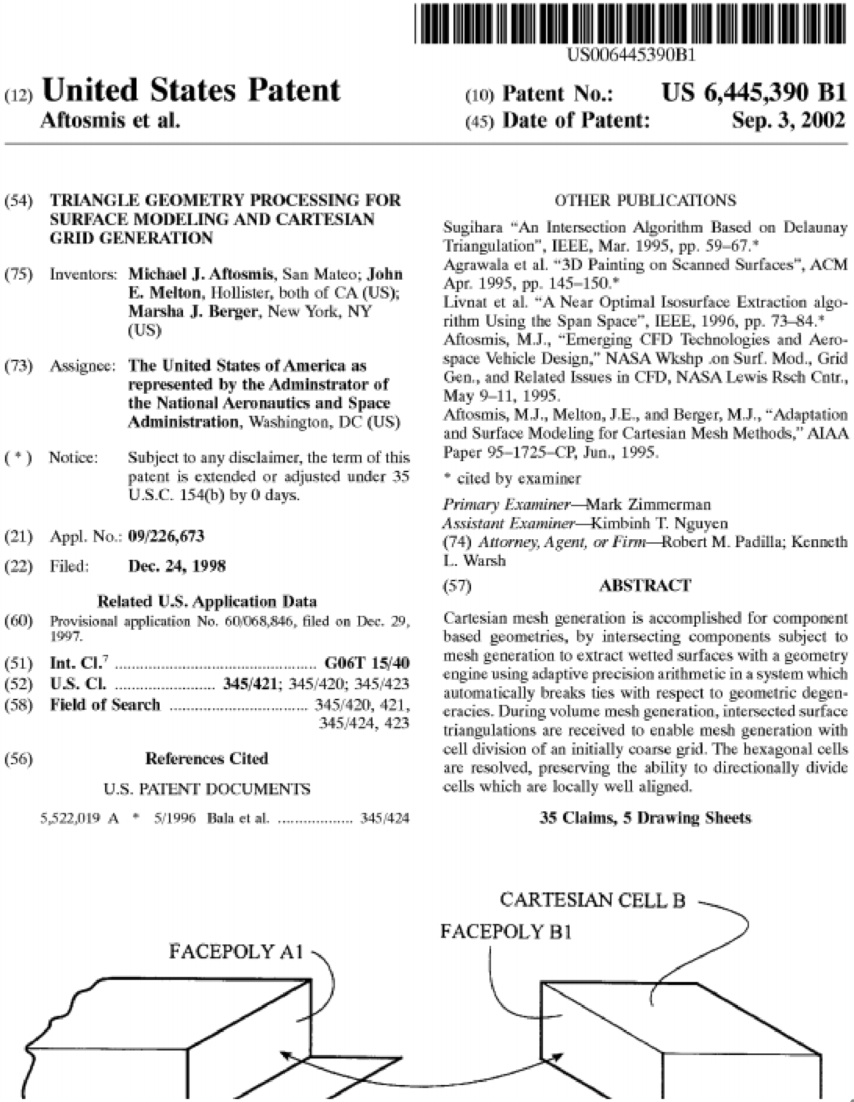
UNIVERSITY OF CAMBRIDGE

Cut cells automatic mesh generation, direct method

- Directly compute the STL-grid intersections.

- Used in Cart3D.

- Details are **very** algorithmically involved. (Aftosmis et al., Robust and efficient Cartesian mesh generation for component-based geometry, AIAA, 1998).



US006445390B1

(12) **United States Patent**
Aftosmis et al.

(10) **Patent No.:** **US 6,445,390 B1**
(45) **Date of Patent:** **Sep. 3, 2002**

(54) **TRIANGLE GEOMETRY PROCESSING FOR SURFACE MODELING AND CARTESIAN GRID GENERATION**

(75) Inventors: **Michael J. Aftosmis**, San Mateo; **John E. Melton**, Hollister, both of CA (US); **Marsha J. Berger**, New York, NY (US)

(73) Assignee: **The United States of America as represented by the Adminstrator of the National Aeronautics and Space Administration**, Washington, DC (US)

(*) Notice: Subject to any disclaimer, the term of this patent is extended or adjusted under 35 U.S.C. 154(b) by 0 days.

(21) Appl. No.: **09/226,673**

(22) Filed: **Dec. 24, 1998**

Related U.S. Application Data

(60) Provisional application No. 60/068,846, filed on Dec. 29, 1997.

(51) **Int. Cl.**[7] **G06T 15/40**
(52) **U.S. Cl.** **345/421**; 345/420; 345/423
(58) **Field of Search** 345/420, 421, 345/424, 423

(56) **References Cited**

U.S. PATENT DOCUMENTS

5,522,019 A * 5/1996 Bala et al. 345/424

OTHER PUBLICATIONS

Sugihara "An Intersection Algorithm Based on Delaunay Triangulation", IEEE, Mar. 1995, pp. 59–67.*
Agrawala et al. "3D Painting on Scanned Surfaces", ACM Apr. 1995, pp. 145–150.*
Livnat et al. "A Near Optimal Isosurface Extraction algorithm Using the Span Space", IEEE, 1996, pp. 73–84.*
Aftosmis, M.J., "Emerging CFD Technologies and Aerospace Vehicle Design," NASA Wkshp .on Surf. Mod., Grid Gen., and Related Issues in CFD, NASA Lewis Rsch Cntr., May 9–11, 1995.
Aftosmis, M.J., Melton, J.E., and Berger, M.J., "Adaptation and Surface Modeling for Cartesian Mesh Methods," AIAA Paper 95–1725–CP, Jun., 1995.

* cited by examiner

Primary Examiner—Mark Zimmerman
Assistant Examiner—Kimbinh T. Nguyen
(74) *Attorney, Agent, or Firm*—Robert M. Padilla; Kenneth L. Warsh

(57) **ABSTRACT**

Cartesian mesh generation is accomplished for component based geometries, by intersecting components subject to mesh generation to extract wetted surfaces with a geometry engine using adaptive precision arithmetic in a system which automatically breaks ties with respect to geometric degeneracies. During volume mesh generation, intersected surface triangulations are received to enable mesh generation with cell division of an initially coarse grid. The hexagonal cells are resolved, preserving the ability to directionally divide cells which are locally well aligned.

35 Claims, 5 Drawing Sheets

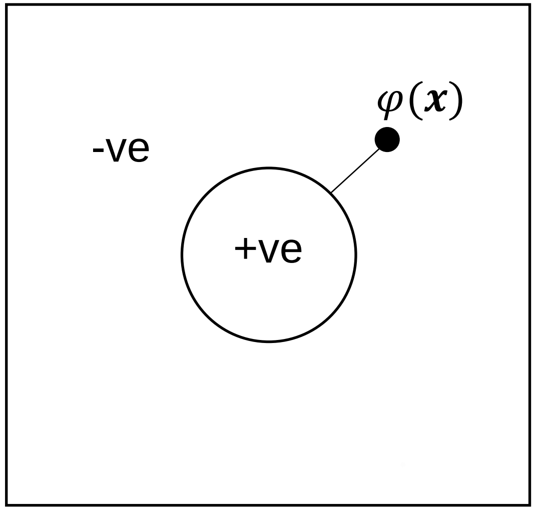
Cut cells automatic mesh generation, level-set method

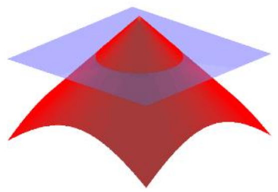
- Level-set methods will be covered in much more detail later. For now, we only need to understand what the zero level set of the signed distance function (SDF), $\varphi(\boldsymbol{x})$, represents.

- $\varphi(\boldsymbol{x})$ at a point $\boldsymbol{x}$ is the shortest distance from x to the geometry, with the sign determined by whether x is inside the geometry.

- The zero level set (zero crossing) of $\varphi(\boldsymbol{x})$ identifies the boundary of the geometry.

- Several fast methods exist to compute the SDF in 2D and 3D.
  (e.g. Mauch, A fast algorithm for computing the closest point and distance transform, Caltech Tech. Report, 2000).



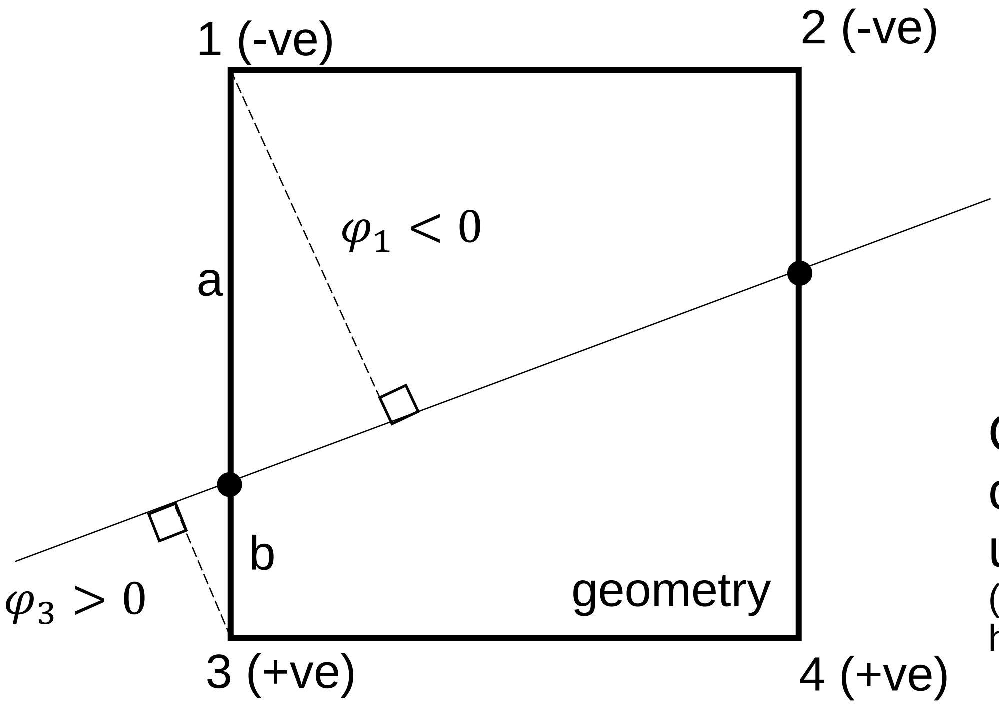


https://en.wikipedia.org/wiki/Signed_distance_function

UNIVERSITY OF CAMBRIDGE

Calculating cut cell geometric parameters from $\varphi(x)$

- The value of the signed distance function at cell vertices can be used to linearly reconstruct the interface in cut cells.

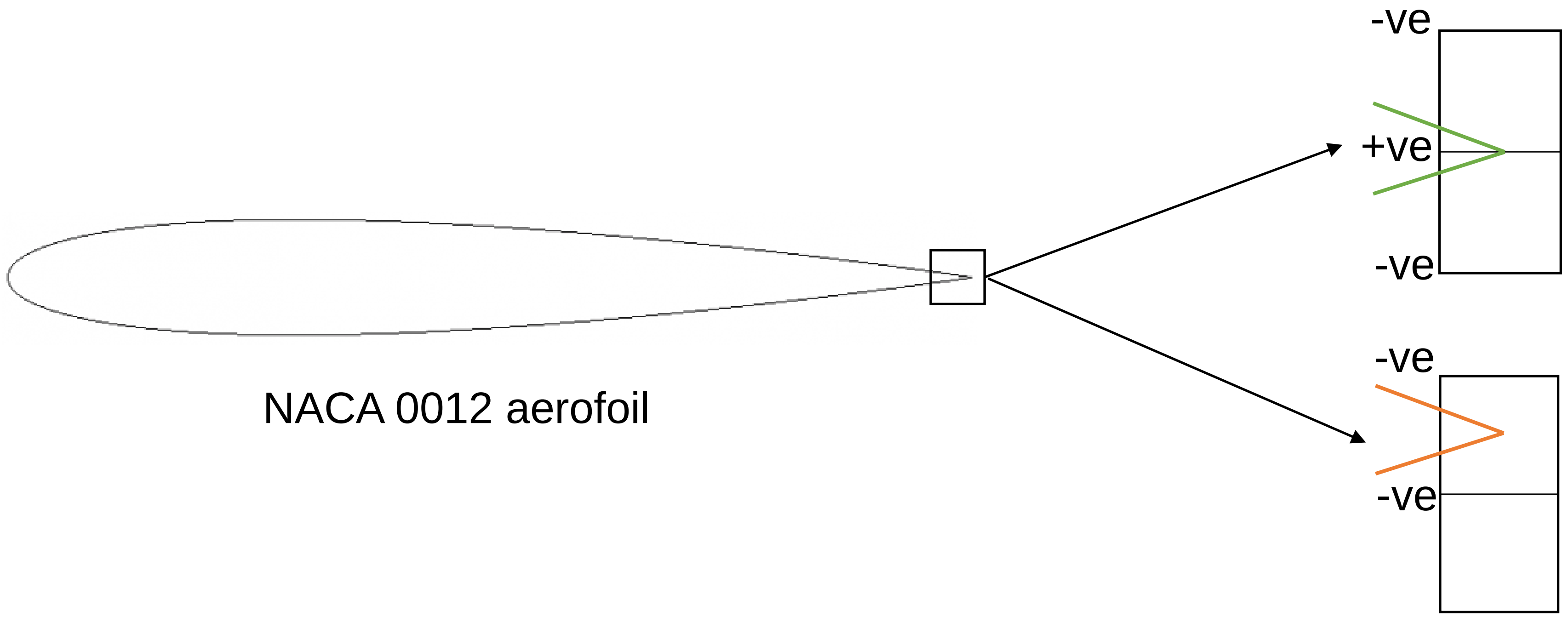


$$\frac{|\varphi_1|}{|\varphi_2|} = \frac{a}{b}$$

Once the intersection points have been found, the cut cell geometric parameters can be calculated using standard polygon/polyhedron formulas.
(e.g. Gokhale et al., A dimensionally split Cartesian cut cell method for hyperbolic conservation laws, J. Comput. Phys, 2018)

UNIVERSITY OF CAMBRIDGE

SDF based mesh generation challenges: Thin features

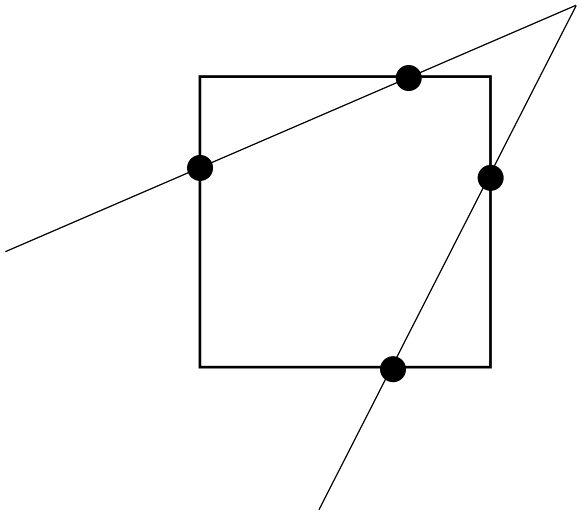


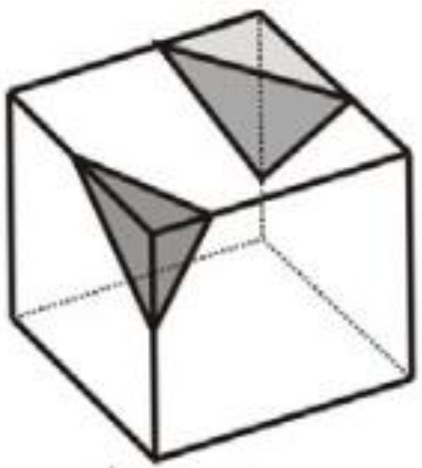
NACA 0012 aerofoil

- Features thinner than the mesh resolution may not be captured by the interface reconstruction.
- This can be solved by using targeted increased mesh resolution where required (AMR can be very useful for this).

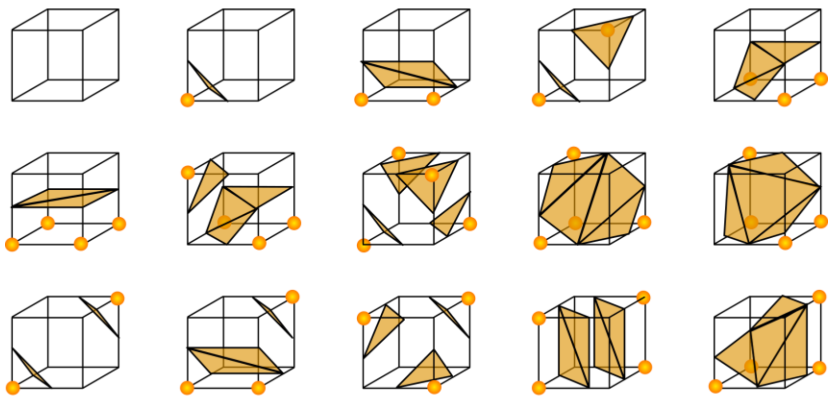
UNIVERSITY OF CAMBRIDGE

SDF based mesh generation challenges: Multiply-cut cells





Meinke et al., A cut-cell method for sharp moving boundaries in Cartesian grids, Computers & Fluids, 2013.



https://en.wikipedia.org/wiki/Marching_cubes#/media/File:MarchingCubes.svg

- This problem is difficult to bypass with just mesh refinement (the computational cost exceeds greatly). Ideally, we need additional computational logic to deal with these cases.
- Alternatively, an algorithm like ‘Marching Cubes’ [1] can be used. This was originally designed for CT and MRI scans, and maps the vertex SDF configuration to 2^8 pre-computed cases based on the signs of the SDF.

UNIVERSITY OF CAMBRIDGE

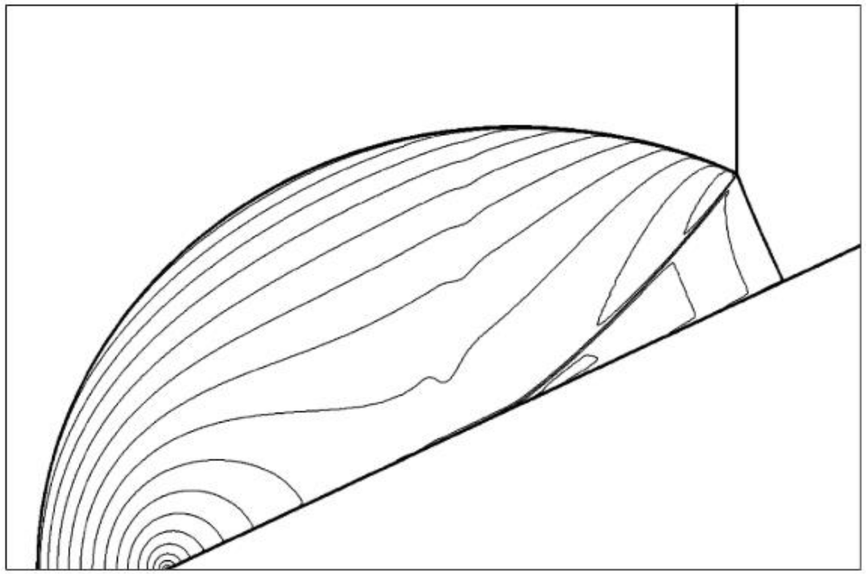
[1] Lorensen, W.E. and Cline, H.E., Marching cubes: A high resolution 3D surface construction algorithm, SIGGRAPH, 1987.

Cut cell simulation examples

UNIVERSITY OF CAMBRIDGE

M=1.7 shock reflection from a wedge

Experimental shadowgraph



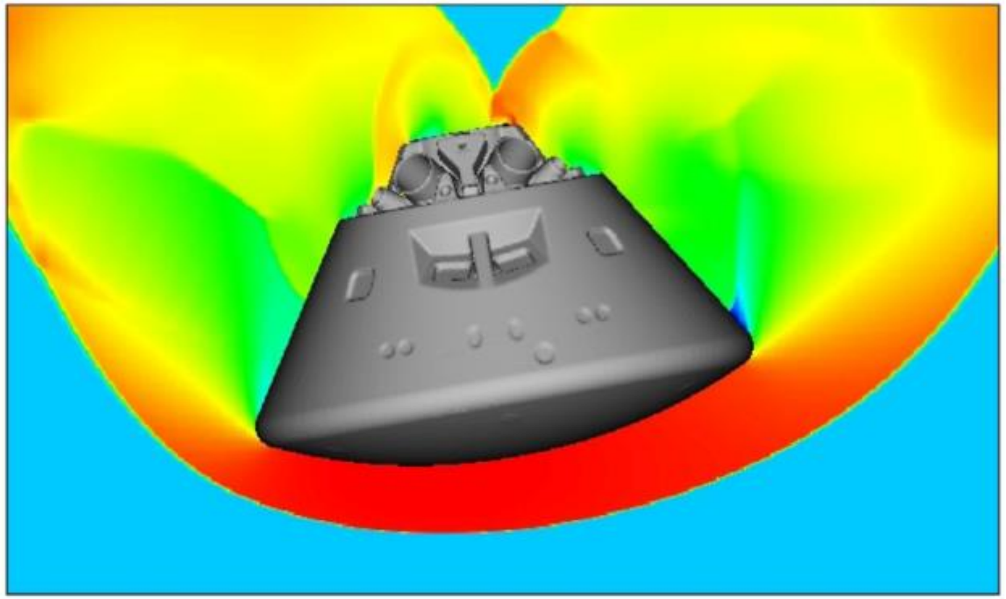
Numerical density contours

Gokhale et al., A dimensionally split Cartesian cut cell method for hyperbolic conservation laws, J. Comput. Phys, 2018

UNIVERSITY OF CAMBRIDGE

M=20 inviscid flow over a re-entry vehicle

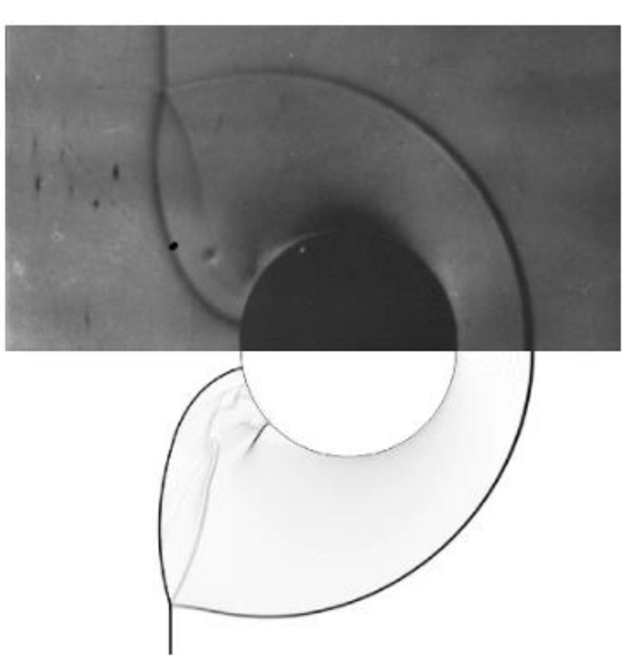
Reconstructed cut-cell geometry

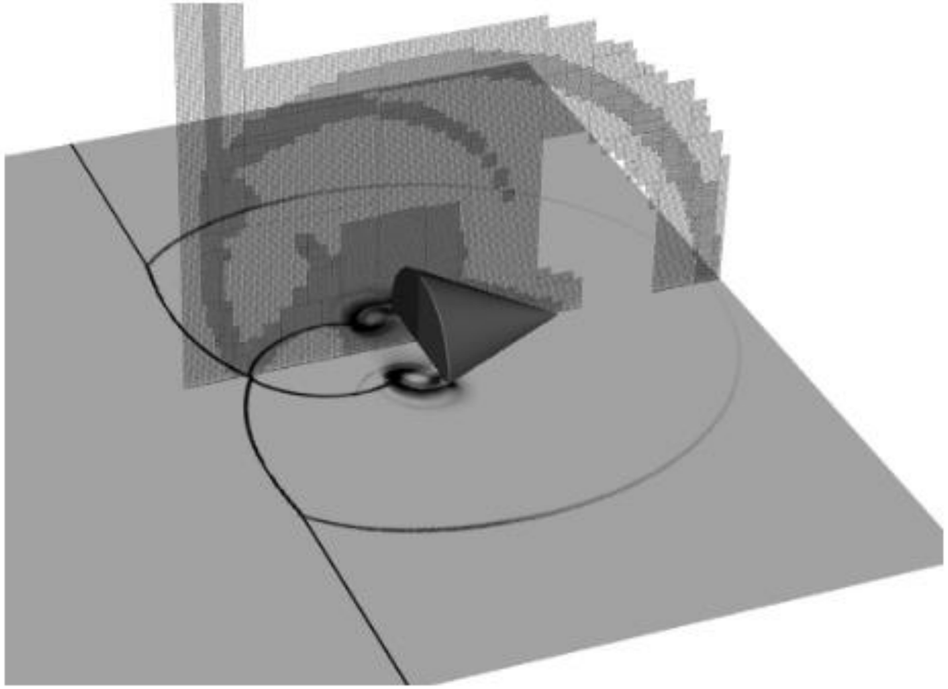


Numerical pressure contours

Gokhale et al., A dimensionally split Cartesian cut cell method for hyperbolic conservation laws, J. Comput. Phys, 2018

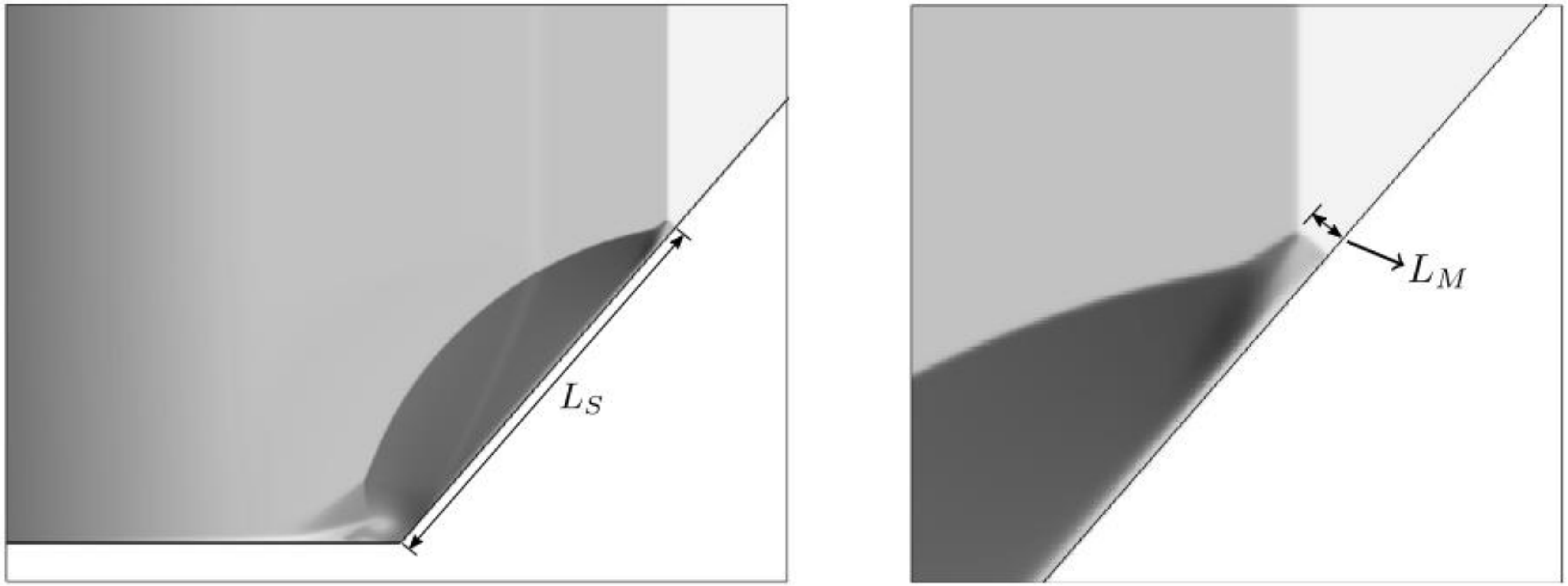
Inviscid moving geometries





Bennett et al., A moving boundary flux stabilization method for Cartesian cut-cell grids using directional operator splitting, J. Comput. Phys, 2018

M=7.1 shock boundary layer interaction

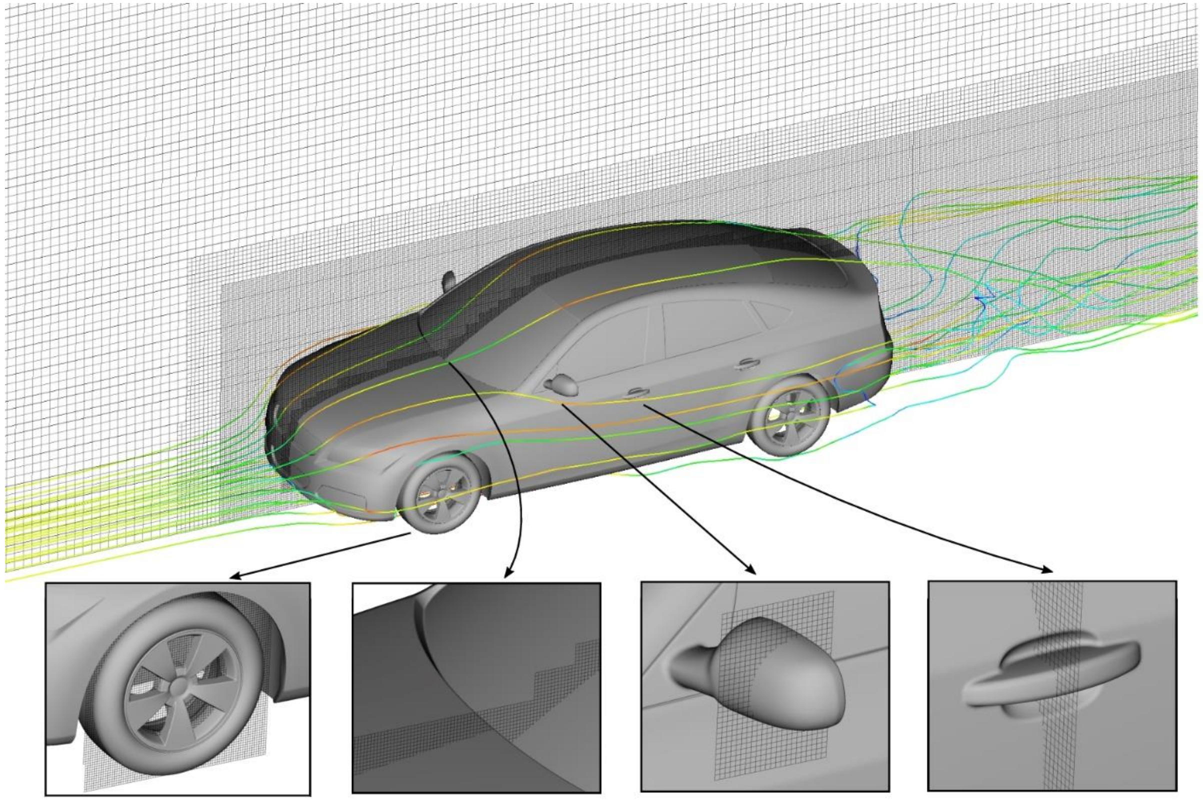


Shock boundary layer interaction on the bottom surface leads to separation bubble creation followed by re-attachment

Gokhale et al., A dimensionally split Cartesian cut cell method for the compressible Navier-Stokes equations, J. Comput. Phys, 2018

UNIVERSITY OF CAMBRIDGE

40 m/s ($Re_H = 1.48 \times 10^6$) flow over the DrivAer model

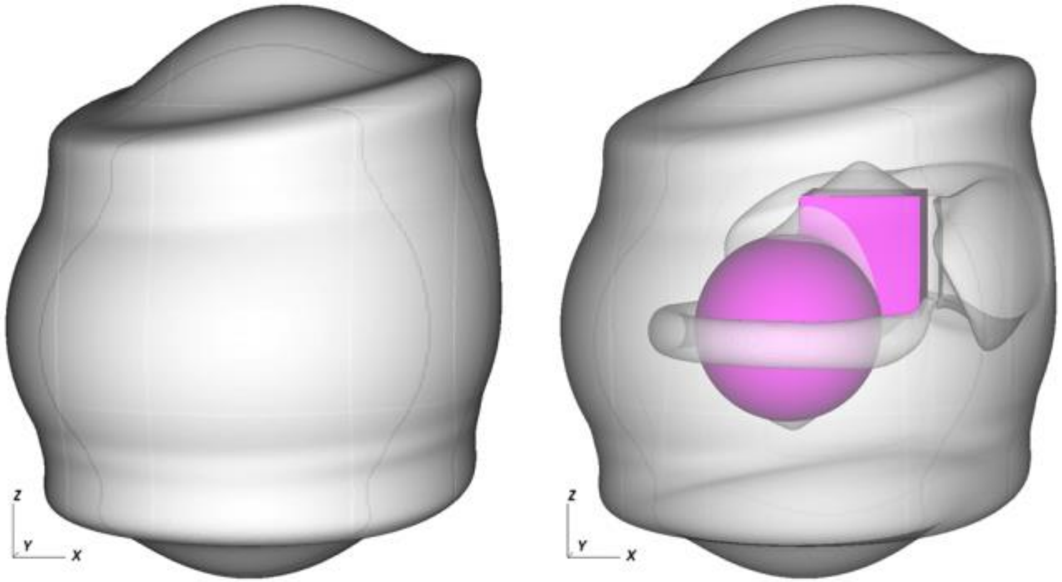


Numerical streamlines

Gokhale, N.G., A dimensionally split Cartesian cut cell method for the Computational Fluid Dynamics, PhD Thesis, 2018

UNIVERSITY OF CAMBRIDGE

3D incompressible flow of a complex geometry in a Bingham fluid (viscoplastic, non-Newtonian)



Numerical yield surface

Sverdrup, K. et al., An embedded boundary approach for efficient simulations of viscoplastic fluids in three dimensions. Phys. Fluids, 2019.

UNIVERSITY OF CAMBRIDGE

Summary and outlook

This lecture covered the following topics:

- An overview of the mesh generation methods used in continuum modelling.
- An introduction to modern Cartesian-based methods, with a focus on the Cartesian cut-cell method.
- Level-set based automatic cut cell mesh generation.
- Cut cell simulation examples.

Next time:

- How cut cell methods solve the small cell problem.
- The mathematics behind the operation of one method.

UNIVERSITY OF CAMBRIDGE