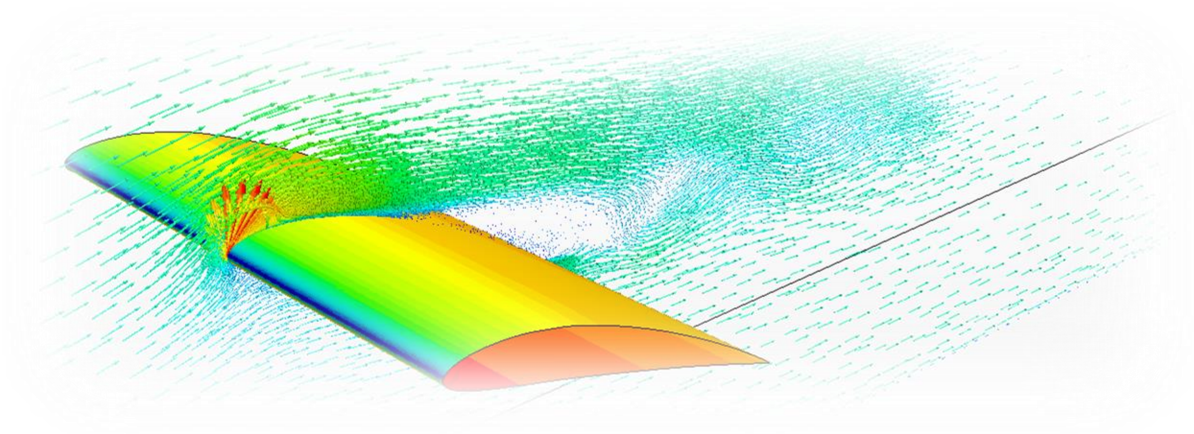
CCM+ Model creation/running workflow summary

Laurence Marks

November 2023

This document summarises the workflow used to create a simple CFD model in CMM+

	Operation	Notes
1	Create new simulation file	Sets up model including # of cores used
2	Import step file	Read model geometry
3	Split by patch	Creates sets used to define boundary conditions
4	Assign parts to regions	Sets up relationship between solids and surfaces
5	Set BC region types	Defines types of BC's (wall, inlet, outlet etc)
6	Generate mesh	Creates CFD mesh
7	Create mesh scene	Allows visualisation of mesh
8	Select physics models	Sets solution parameters (properties, turbulence etc)
9	Run	Runs the model
10	Monitor solution	Plot residuals and scalar scene whilst solve progresses
11	Post process	Plot results



Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com