ES440 Computational Fluid Dynamics

Dr Yongmann M. Chung
School of Engineering, University of Warwick, UK
September 2012

Introduction to Star-CCM+ software – Session 5

Fluid and Solid

Introduction

As shown in the pipe flow and bump simulation, many CFD simulations are that of just a fluid continuum with boundary conditions (walls) on the sides e.g pipe flow. However in many engineering situations the flow around an object (solid continuum) would like to be analysed, for instance; an aircraft wing, car or in the case of Adidas engineers even a football. Below is a guide on how to convert your model of your solid continuum in this case an aerofoil so that it is suitable for Star-CCM. If you were making your own simulation from scratch, model your part, combine it so that it is one solid body and then draw a domain around your part (solid continuum), this domain then representing your fluid continuum and then follow the guide below.

- Download the **aerofoil.sldprt** from the ES440/ES911 website then open in Solidworks.
- You will see a rectangular block and two solid bodies labelled in the parts tree.
- Click for hidden lines visible in the Display style tab and you should still see just a rectangular box.
- The next step is then to right click > **edit** feature on the domain solid body in the parts tree.
- Un-tick the merge result and click the green tick, you should now see the aerofoil within the box.
- The next step is then to click >**insert>features>combine**
- In the options then select the domain as the main body and the aerofoil as the part to be subtracted, click ok and you will now have the domain minus the aerofoil (check this with a section view if you desire)

In essence, the Solidworks model is now that of the fluid around the aerofoil, save the file as a parasolid ***.x_t** file and load on to Star CCM in the same manner as you did for the bump simulation.