THE BENEFIT OF ANSA TOOLS IN THE DALLARA CFD PROCESS

Simona Invernizzi,
Dallara Engineering, Italy,

KEYWORDS
automatic tools, batch mesh, DFM, morphing, ride height maps

ABSTRACT

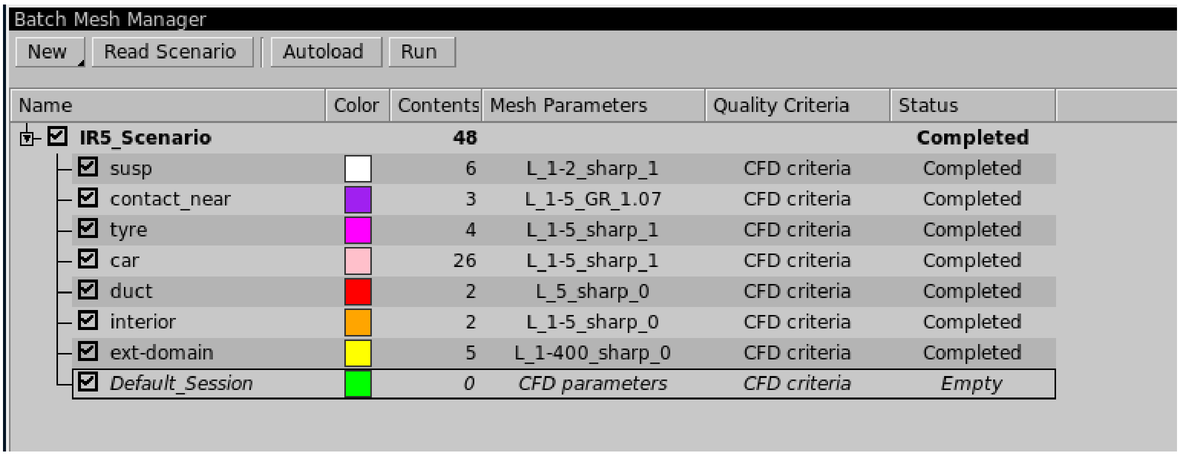
In the last few years, Dallara has seen a strong increase in external consultancies. The Customer's needs and satisfactions are our target, therefore our CFD engineers have to work with different types of cars, from high performance road cars to formula and racing cars and face a variety of problems. In order to be competitive in the Motorsport world it is essential to speed up different phases of the CFD process, saving human time and preserving the accuracy and the reliability of the results. The present paper aims at highlighting the key role that the Ansa "mesher" has played in our process by increasing the flexibility of working on the topology and reducing the human time spent for the surface mesh generation, using the "batch mesh mode".
Furthermore, the DFM tool has given Dallara the opportunity to reproduce, through CFD, automatic ride height maps as is done in wind tunnel testing. By changing the ride heights it is possible to analyse the car behaviour on a track in different configurations during the entire lap, from straight lines to cornering with also the assessment of the pitch sensitivity during braking manoeuvres. The human time has been reduced dramatically with a fully automated process based on Ansa scripts and the CFD Engineers are able to investigate the effect of aerodynamic solutions at different configurations of the car. Finally, these tools have increased the efficiency of the whole CFD process, boosting our competitiveness and throughput.

INTRODUCTION

In the whole Dallara CFD process the 60% of the human time is spent in CAE modelling, the 35% is dedicated in analysing the results, while the remaining 5% is machine time.
In order to invest the precious human time in developing new aerodynamic solutions and satisfy Customers requests, Dallara CFD-R&D department, in collaboration with Beta CAE support, tested and customized different Ansa tools, saving modelling time.
Moreover, the robustness and the flexibility of Ansa, in managing topology and mesh, allow our CFD people to face with complex database, formula cars as well as road cars, and to realize particular models, such as required from challenging simulations, like the under-hood thermal analysis.
The tools implemented with Ansa scripts are the "Batch mode" and the "DFM utility". The first one gains human time during the pure meshing phase, while the DFM acts on the whole CAE modelling, saving also CAD hours to reproduce a different car set up.

1. BATCH MESH MODE

The batch mesh is a powerful tool to automatically create the surface mesh and the volume mesh for a complete CFD model. This tool is applied to speed up the surface mesh generation, preserving high quality mesh thanks to dedicated quality options, and providing numerical solutions consistent with the ones achieved with the standard meshing process. The idea is to create a specific template, customized according to the model in terms of number and attributes. This template leads to flexibility since, once created, it can be exported and used as much time as needed.
The batch mesh creates a meshing scenario, formed by sessions, where a session can be defined as a group of PID, sharing the same mesh parameters and the same quality criteria.
The algorithm automates the surface mesh generation, that for complex geometries, race and high performance road car, is not easy at all: in these cases, a visible look to the resulting mesh is essential.



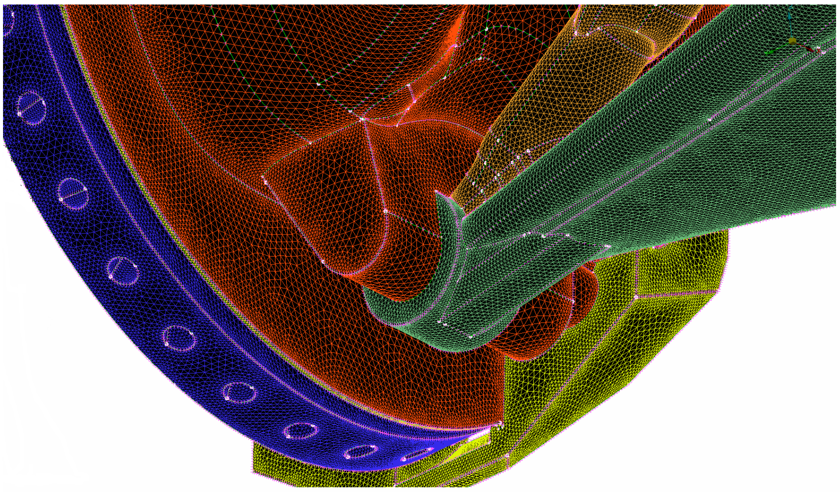
Name	Color	Contents	Mesh Parameters	Quality Criteria	Status
IR5_Scenario		**48**			**Completed**
susp		6	L_1-2_sharp_1	CFD criteria	Completed
contact_near		3	L_1-5_GR_1.07	CFD criteria	Completed
tyre		4	L_1-5_sharp_1	CFD criteria	Completed
car		26	L_1-5_sharp_1	CFD criteria	Completed
duct		2	L_5_sharp_0	CFD criteria	Completed
interior		2	L_1-5_sharp_0	CFD criteria	Completed
ext-domain		5	L_1-400_sharp_0	CFD criteria	Completed
Default_Session		*0*	*CFD parameters*	*CFD criteria*	*Empty*

(Fig.1: Batch mesh manager)

The human time saved with the batch mesh is up to 75% on the pure surface mesh generation, and up to 50% on the whole surface phase.
Obviously, the statistics depend on the model complexity: the following table refers to a Dallara open wheel car, used for testing the batch mesh.

	STANDARD MESHING		BATCH MESHING
TOPO & PID	**4 H**		**4 H**
SURFACE MESH	**8 H**	75% saved →	**2 H**
TOTAL SURF. MESH	**12 H**	50% saved →	**6 H**

Regarding the volume mesh, achieved starting from the surface one with a bottom-up approach, it can be noticed a slightly worst volume quality of 1% in the average cell volume, for very complex geometries.
The delta in terms of forces are of the standard deviation order, and therefore can be classified as a part of the numerical uncertainty of the simulation.



(Fig.2: Batch mesh results)

2. A CUSTOMIZED TOOL FOR AUTOMATIC RIDE HEIGHT MAP

"Driving a car as fast as possible (in a race) is all about maintaining the highest possible acceleration level in the appropriate direction."

The aerodynamic development of a racing car has to consider the behavior of the car around a whole lap: analyzing how the aero performance changes on the straights, cornering and braking maneuvers. The pitch sensitivity analysis is mandatory to understand the aerodynamic potential of a car on the track and how to achieve the target in different configurations. For this reason, during wind tunnel sessions, the tests are conducted at different configurations (pitch, yaw, heave, roll and steer) thanks to a fully automated model motion system. Moreover, with the CAM (continuous acquisition motion) testing capability it is possible to continuously change the configuration of the car simulating the whole lap, without doing discrete data acquisitions: for these features the wind tunnel represents the most efficient tool to perform pitch sensitivity analysis. However, in the last few years, the involvement of CFD simulation in racing car projects has been pushed so much that nowadays different racing cars are completely developed only by using CFD, reducing time and costs.
For this reason the CFD-R&D department has put a great deal of effort in improving its service, developing different tools to speed up and automate some phases of the process, preserving the high quality of the results and products. In this context we can introduce the "Ride height map" tool, based on the Ansa DFM utility, implemented in collaboration with Beta CAE support. This tool allows to automatically change the ride height of the car, saving human modelling time (CAD and meshing time) and preserving the quality of the numerical solution, within a certain range of ride heights, within which the mesh deformation is acceptable. This solution allows CFD engineers to perform the aerodynamic map, as is done in wind tunnel, starting from one or more baseline configurations.
The results of the sensitivity analysis conducted with CFD simulation, using the DFM tool to change the set up configuration, correlate well with those achieved in the wind tunnel, using the latter as benchmark for CFD simulations.

What follows is the typical aero map (derived from the CAM) of an open wheel car developed in Dallara, both using wind tunnel and CFD tools: we have considered this database in order to validate the DFM tool by Ansa. From the graph below, it can be noticed that the aero map points are quite a lot: using the DFM tool we cannot cover all the map like in the wind tunnel, and even for a CFD sub-map two further baselines are necessary. This means that more than one reference set up (in terms of front and rear ride heights) has to be modeled in CAD, in order to use DFM tool for simulating the cloud of points of interest.

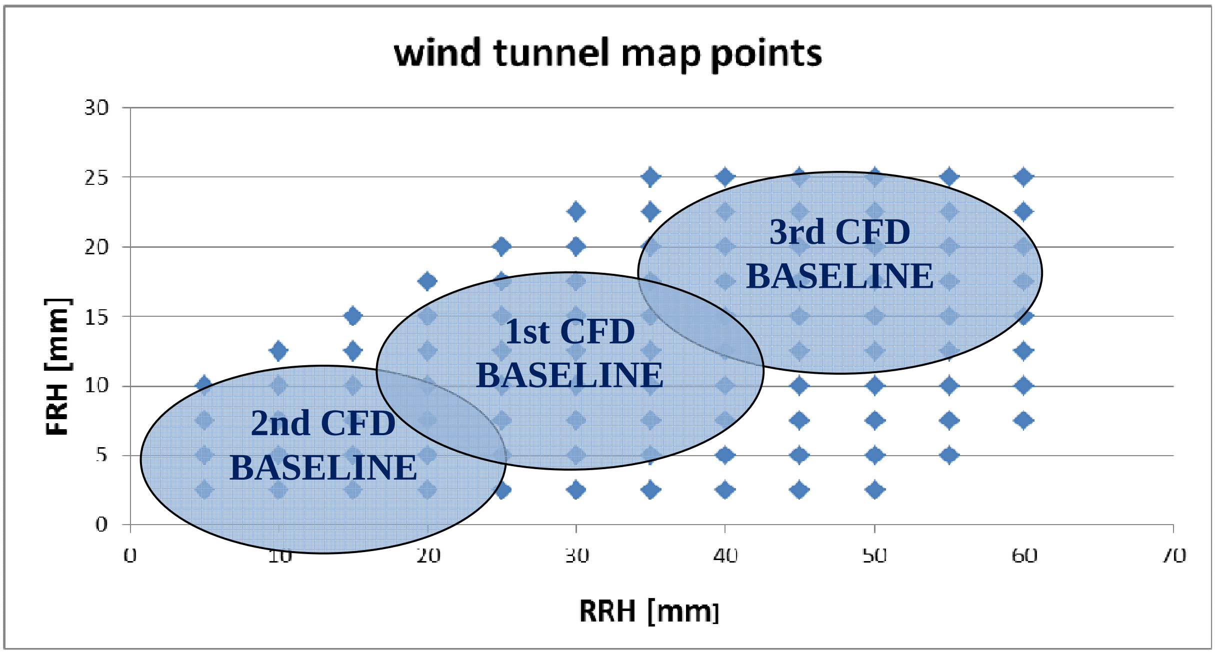


Fig.3: Wind tunnel map points from CAM

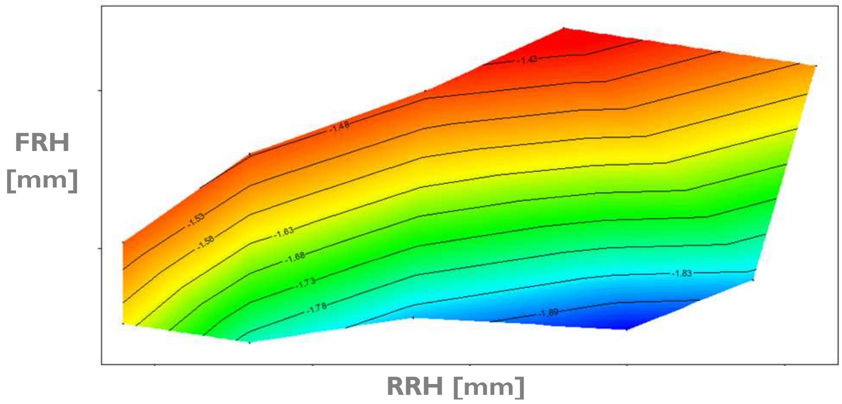


Fig.4: Example of an Aero-load distribution on front and rear ride heights

The DFM utility concept applied to the ride height change has been written as a set of functions into a *batch* script by BETA-CAE support and then collected, rearranged and completed into a Dallara-made script. Moreover, to completely fulfill the Dallara targets a home-made graphical interface has been implemented, in order to avoid opening Ansa, thus saving a significant amount of human time.

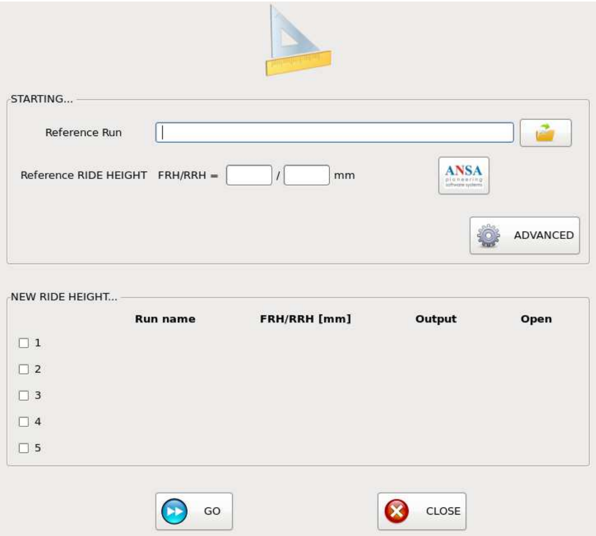
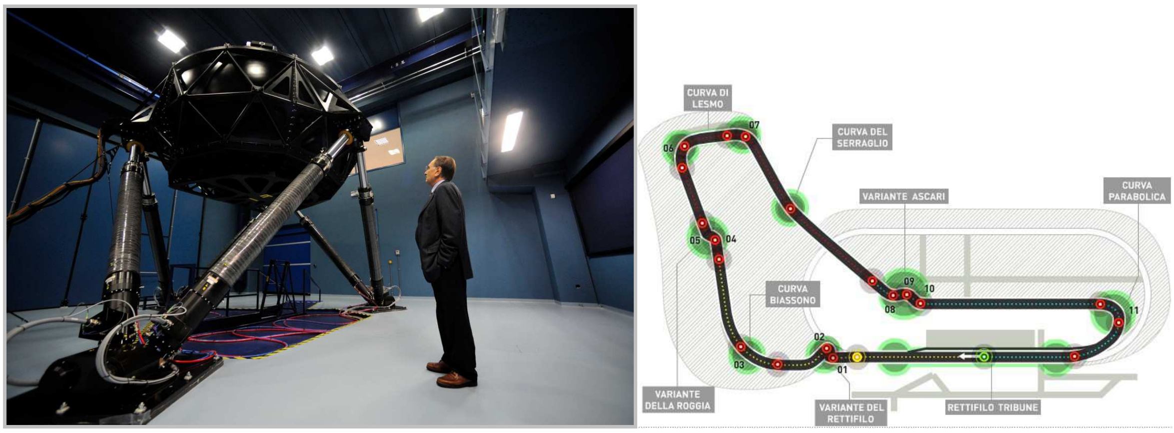


Fig.5: DALLARA-made GUI

Ride height set up	Human hours
STANDARD PROCESS **(CAD+MESH)**	**16 h (10h + 6h)**
ANSA DFM + GUI	**0 h**

Once the pitch sensitivity analysis is available, either with wind tunnel or CFD, a Dallara-made tool, developed as a results of interdisciplinary collaboration of aerodynamic and vehicle dynamic departments, starting from the previous sensitivity analysis, is used to highlight if there are any criticalities for the car.
After the aerodynamic development is completed, the professional Dallara driving simulator (the man in the loop), can be used to simulate and predict the behavior of the car on a specific track, collecting precious feedback even before the track tests.



(Fig.6: Dallara driving simulator (2012): from pitch sensitivity to track prediction)

CONCLUSIONS

Thanks to Ansa tools and BETA CAE support in developing new methodologies and strategies, Dallara has been able to push the meshing phase of the CFD process and save human time in order to reemploy it in the aerodynamic development of a project. Therefore, the entire process takes advantage from this situation: increasing our competitiveness and our capability to satisfy customer needs.