Flow Analyses for Spacecraft

J. Steelant , J.A. Romera-Perez
Thermal and Environmental Control Section, ESA Directorate of Technical and Operational Support, ESTEC, Noordwijk, The Netherlands

F. Bouckaert, P. Tamburini
ATV Team, ESA Directorate of Manned Spaceflight and Microgravity, Les Mureaux, France

J. Witt
Columbus Team, ESA Directorate of Manned Spaceflight and Microgravity, ESTEC, Noordwijk, The Netherlands

The design and construction of International Space Station elements such as Columbus, the Automated Transfer Vehicle and the Cupola are inherently different from those of classical spacecraft due largely to the need for pressurisation of the structure. Air revitalisation in the crew areas and prevention of the accumulation of pockets of dangerous gases in stagnant regions require a well-controlled flow field, for both safety and comfort. Conductive and radiative heat fluxes are then no longer the only heat-flow mechanisms and heat transfer by forced convection also needs to be taken into account*. The characteristics of this type of flow can either be derived analytically or obtained via empirical correlations. Flow patterns in large volumes such as crew cabins are far from homogenous, which makes mass and heat flow prediction very difficult. One therefore needs to resort to an experimental investigation of the flow field or to the numerical solution of the flow equations, usually referred to as Computational Fluid Dynamics (CFD).

An experimental set-up is essential for validating and fine-tuning the correlations used in flow-analysis tools. A numerical tool, however, allows the designer to understand the physical behaviour of flows and to optimise the final design by setting up and studying different configurations more easily and more cheaply. Consequently, the Agency decided to extend its existing thermal tools with a CFD tool, i.e. CFD-RC, in order to address convection-dominated heat-transfer problems in space. At the same time, the outcome of a CFD analysis can also be used to fine-tune the classical tools, such as ESATAN, thereby exploiting the strengths of all available approaches. CFD is now being used extensively for ventilation and heat-transfer analyses for the pressurised module of the Automated Transfer Vehicle (ATV). Another application is to the venting of payload chambers in the Columbus module, which could perturb the Space Station's microgravity environment.

* Although convection is also present in unpressurised spacecraft, it is mainly limited to fluid flows in tubes or other confined geometries, for which the flow pattern is generally known.

CFD: a versatile tool for multi-disciplinary analyses

To generate the output of a flow analysis (e.g. forces, heat fluxes, mass flows and concentrations) efficiently, a whole battery of software and hardware tools is required. The following paragraphs attempt to describe the different steps and needs in the 'computational' process to analyse engineering problems in this area.

Pre-processing: mesh generation

Pre-processing consists of defining the geometrical description of the to-be-studied model and the discretization of the two- or three-dimensional domain within or around the model. Contrary to most thermal analyses, where the calculation of conductive and/or radiative heat fluxes only requires surface discretization of the model geometry, a fluid-dynamic computation also requires the volume in or around the geometry to be discretized. Hence, the number of computational nodes needed increases very rapidly with the detail in the model. For 3D-geometries, this easily reaches from 100 000 to 1 000 000 nodes and even higher. This vast number of nodes, along with the description of complex geometries, necessitates the use of a powerful mesh-generation tool that allows the user to produce a computational mesh in a (semi)-automatic way. Although these tools are commercially available, the mesh generation might still require quite a lot of effort. The effort largely depends on the requested grid quality, the mesh type (structured vs. unstructured) and the geometrical complexity.

In general, unstructured meshes (Fig. 1a) can be easily generated independently of the geometrical complexity, both in 2D and 3D. Although structured meshes (Fig. 1b) are to be preferred for reasons of accuracy in cases of aligned flow, e.g. boundary or shear layers, their generation can sometimes be difficult and cumbersome. Owing to their nature, they generally tend to generate more points than in the unstructured case. The possibility of constructing a hybrid grid (Fig. 1c), to combine the advantages of both a structured mesh (accuracy) and an unstructured grid (fewer points and easier to produce), is a valuable asset to have within a mesh-generation tool.

Solvers: incompressible vs. compressible

To evaluate the velocity, temperature or pressure field, the highly non-linear Navier-Stokes equations need to be solved. In the past, many techniques for solving the flow equations numerically have been studied. The majority of them can be divided into two major schools: the density-based and the pressure-based methods. The latter method originated from techniques elaborated for the incompressible limit of the flow equations. In this case the density hardly varies, as is the case for example for gases at low speed or liquids (water, oil, etc.). Once these techniques were well-developed and understood, improvements and extensions were introduced to handle also compressible flows up to supersonic speeds.

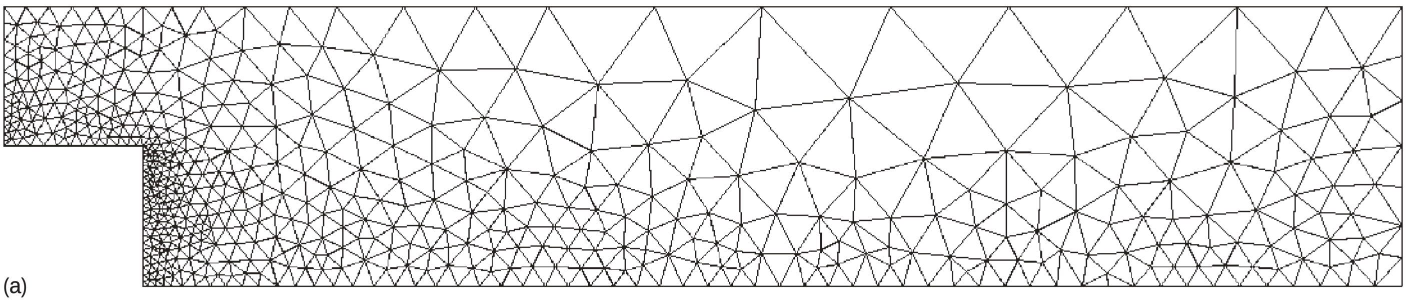
The density-based methods were initiated to tackle problems where compressibility of the fluid is an essential feature. The variability of the density is used to iterate towards a solution. Methods of this kind are generally employed by the aeronautical and aerospace community, where high speeds, compressibility and shocks are common features dominating trans-, super- or hypersonic flows. Here also, however, once the techniques reached maturity, the need emerged to extend this approach towards the incompressible limit.

Nowadays, the commercially available CFD-codes generally follow one of the schools, inspired by their original customers. Only a few codes incorporate both schools into their package. This extra possibility would allow the user to choose the appropriate technique for his/her particular flow problem.

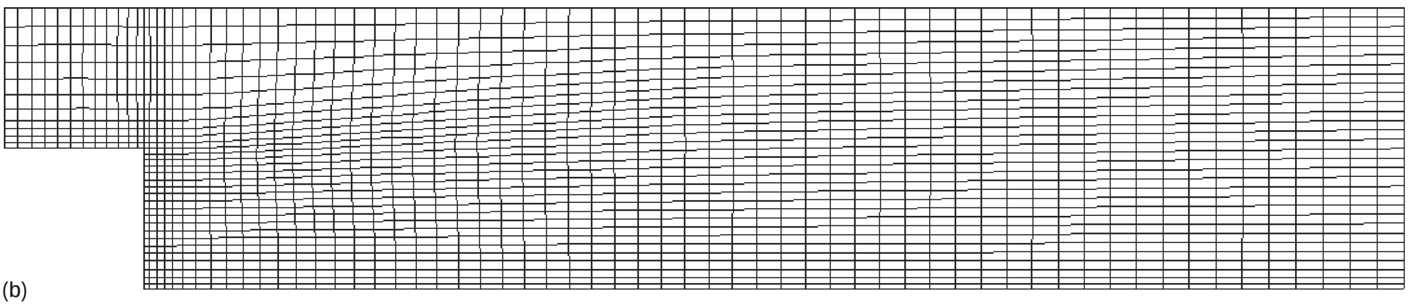
Multi-disciplinarity: the way to fully coupled solutions

In parallel with the evolution of numerical techniques, the need to extend the flow-analysis tools to closely related disciplines or to completely different fields quickly emerged.

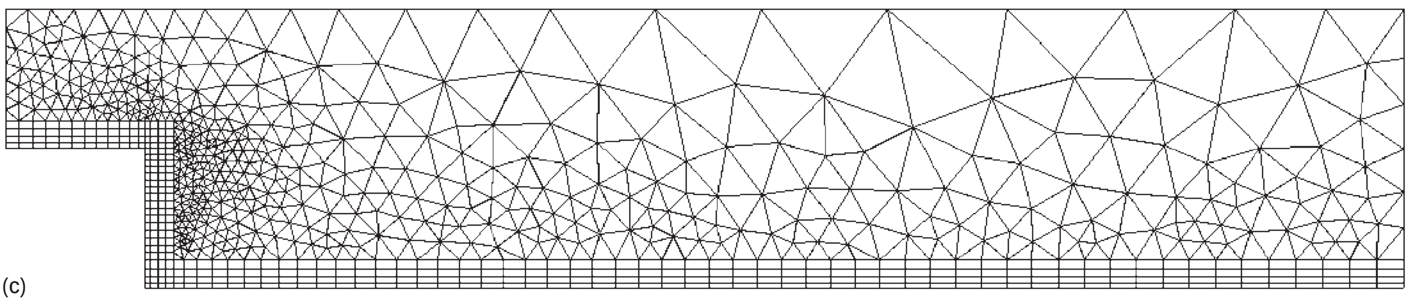
Figure 1. Different mesh-types for sudden tube enlargements:
(a) unstructured mesh with 579 nodes,
(b) structured mesh with 2044 nodes,
(c) hybrid mesh with 1002 nodes



(a)



(b)



(c)

Species transport, chemistry, combustion, two-phase flows, free surfaces, radiation, etc. coupled with the flow equations are some of the topics that were gradually investigated once CFD-techniques became more established. Also the coupling of CFD with heat propagation in solids (conjugate heat transfer), the interaction with structures (steady or modal analysis) electro-magnetic interactions, etc. has resulted in a very versatile evaluation tool applicable across various disciplines.

Post-processing: visualisation and interpretation
The large number of nodes together with the numerous variables, e.g. velocity, temperature, pressure, stresses, gas composition, etc., necessitates the aid of a post-processing tool to visualise, to interpret and to manipulate this vast amount of data. Visualisation enables the user to evaluate the computed information rapidly. Interpretation allows an in-depth analysis of the physical behaviour, and can be as simple as plotting contours, but the tools generally also allow mathematical manipulation of the data. As an example, the calculation of vorticity using the velocity components eases the detection of swirl in the flow field, while the use of traces allows visualisation of the path travelled by a small particle. Many other features are available to facilitate the user's interpretation of the computed data.

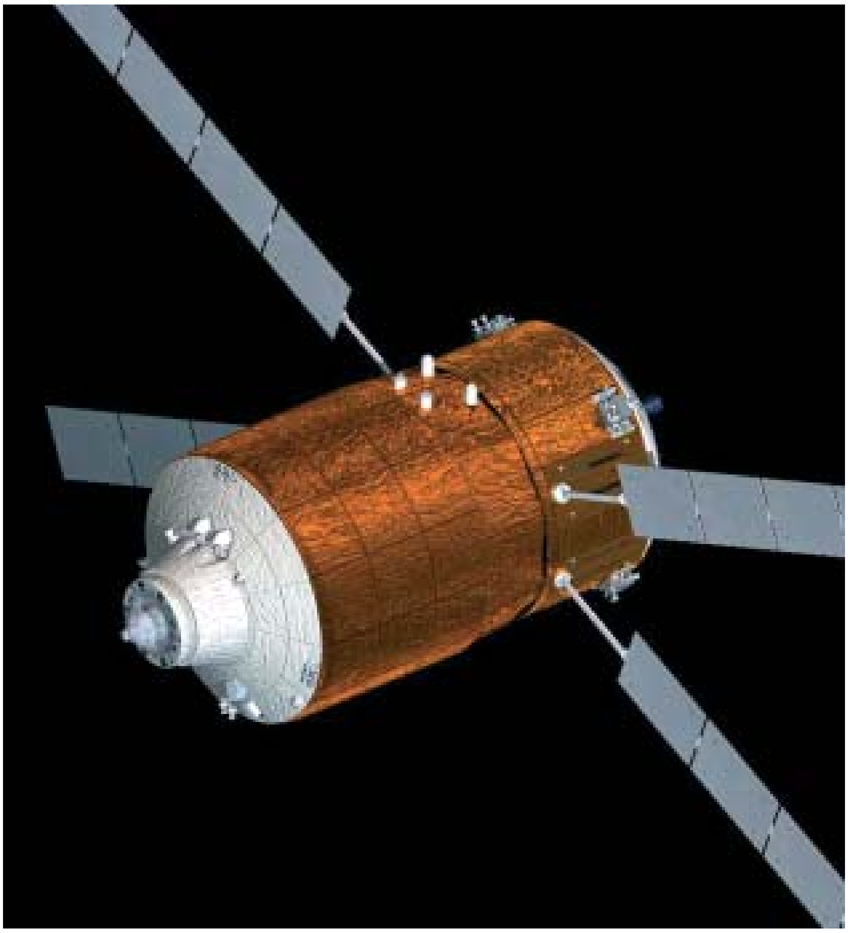


Figure 2. View of the Automated Transfer Vehicle (ATV) with the Integrated Cargo Carrier (ICC) at the front, the propulsion bay at the rear, and the avionics bay in between

Computer science and CFD: harmonisation at its best
Finally, one must not forget that the computational effort itself is an important parameter, especially for three-dimensional problems. This makes CFD a discipline where the relentless increase in computer power enhances its potential. Besides numerical acceleration techniques (multi-grid, pre-conditioning), the use of parallel computers (distributed- or shared-memory computers), the organisation of data-structures (cash hit rates) and communication bandwidths are some of the many hardware aspects to be considered in the efficient coding of CFD-tools. This makes the task both challenging and demanding. The clustering of cheap personal computers to act as a large highly performant computing facility is yet another recent evolution in the continuing quest to accelerate the analysis process.

Applications
ATV: Automated Transfer Vehicle
The low orbital altitude of the International Space Station (ISS) means that it is still susceptible to atmospheric friction, causing it to lose height throughout its lifetime. Hence orbital re-boosting will occasionally be needed to take it back to higher altitude (maximum 460 km). This is one of the tasks foreseen for the ATV, using the propulsion bay located at the rear of the vehicle (Fig. 2). A second task consists of delivering equipment and consumables (food, oxygen, propellant, etc.) to the Station, carried in the vehicle's Integrated Cargo Carrier (ICC) module. This module, located at the front of the ATV, also contains the mechanism needed to dock with the ISS. All dry cargo will be placed in up to 8 racks mounted in the cylindrical portion of the ICC (Fig. 3). The fronts of the racks face a central (2 m x 2 m) corridor, allowing the crew to unload the cargo, whereas their backs form a small air gap with the structural shell. Air revitalisation is provided by blowing air into the ICC (at a rate of 180 m^3/h) from the Russian 'Zvezda' module through a flexible hose.

Hence no advanced ECLS-system is required onboard the ATV itself. Still, to guarantee good ventilation and to ensure an adequate air distribution velocity for crew comfort, a cabin fan mounted in the ICC circulates the air at a rate of 264 m^3/h via appropriately positioned diffusers. Their primary goal is to avoid any stagnant/recirculating flow regions in which toxic gases could accumulate. This might eventually lead to the suffocation of an astronaut, should he/she venture into one of these 'dead-air' zones. The air-conditioning and ventilation is, however, also constrained by

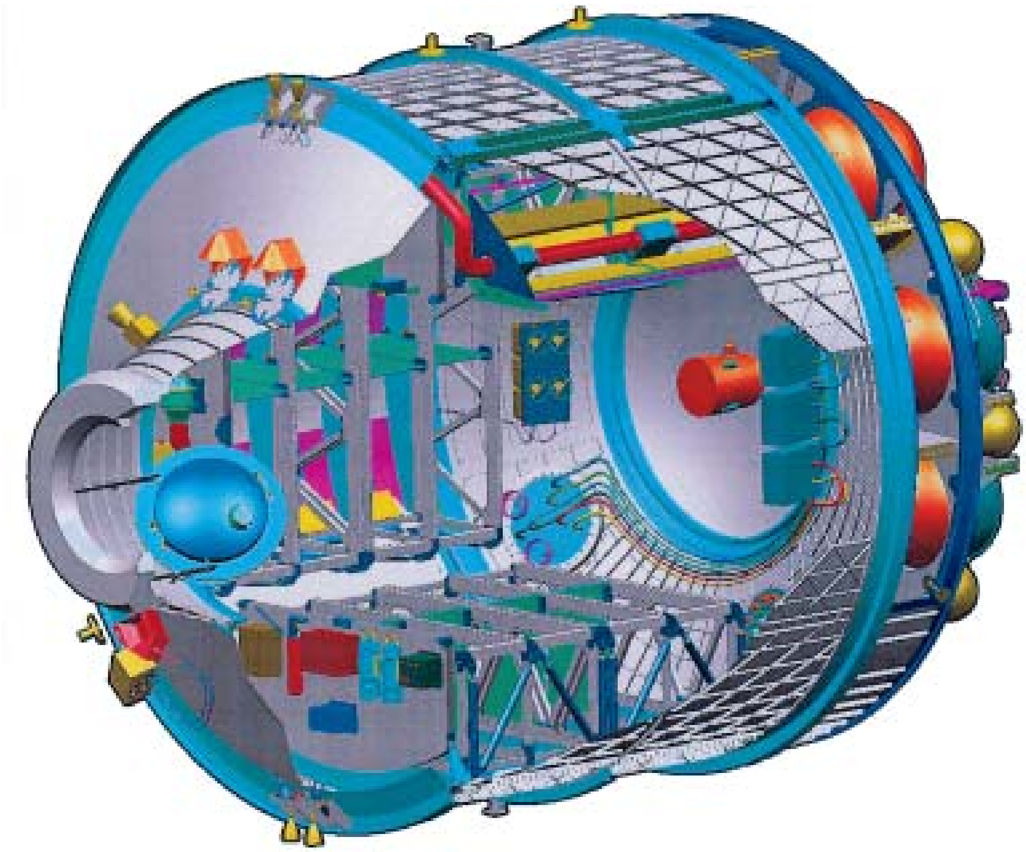


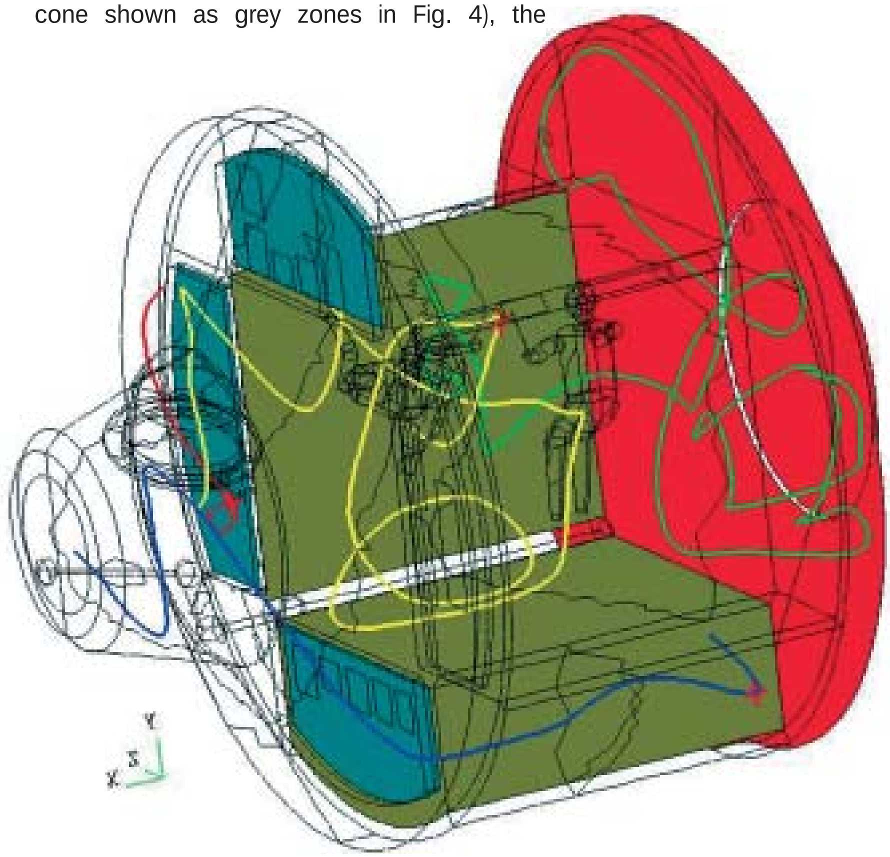
Figure 3. Internal view of the Integrated Cargo Carrier (ICC) with the racks mounted in the centre

certain limits. The air speed should be always lower than 0.25 m/s in the crew cabin, whereas the air temperature should be less than 28°C, but more than the minimum dew temperature of 16°C. Also, the 'touch temperatures' of surfaces exposed to the crew should not exceed 45°C.

The particle traces plotted in Figure 4 are based on the calculated flow field when the ATV is docked to the ISS. The aft diffusers and the flexible hose deliver a high-momentum jet impinging on the rear bulkhead of the ICC, thereby guaranteeing a good washing out of the aft part (green line). The total injected mass flow rate of 240 m^3/h is then deflected towards the front, passing through the crew cabin and the lower stand-offs (blue and green lines). Also four extra cabin diffusers (flow rates of 50 m^3/h each) placed in a staggered way in the upper corners ventilate the central corridor (yellow line). The air flow exiting the cabin and stand-offs is partly deflected towards the main cabin fan sited off-axis in the front cone (red line). The remaining flow (180 m^3/hr) is driven back into the Russian module through the docking port (blue line). Although this visualisation provides a qualitative idea about the flow pattern, it does not provide absolute certainty that every corner is perfectly washed out. To allow a quantitative measurement, therefore, the analytical module is artificially filled up with a trace gas (e.g. CO_2) and then ventilated through the different diffusers with air. The presence of dead/recirculating regions would result in finite concentrations of the trace gas once the ventilation has converged to its final flow field.

Coupled with the flow field, the temperature field is calculated taking into account the heat dissipation of numerous electronic boxes (rectangular areas on the panels in the front cone shown as grey zones in Fig. 4), the

Figure 4. Particle traces inside the pressurised module of the ATV: in the rear (green line), in front of a cabin air diffuser (yellow), in a lower stand-off (blue), and in front of the cabin main fan (red)



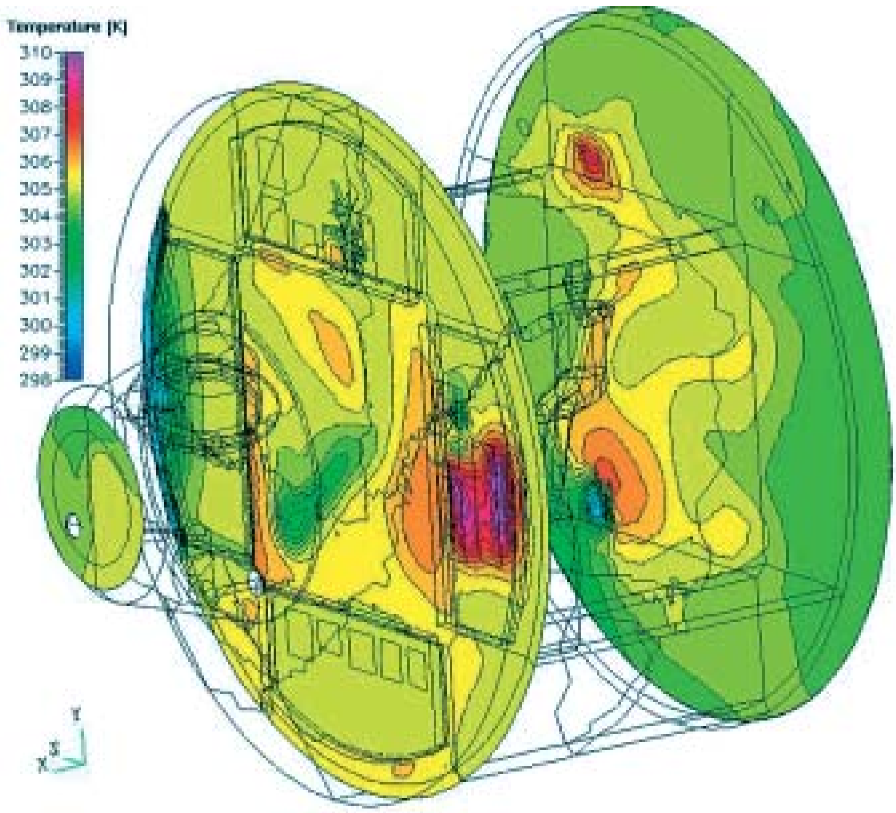


Figure 5. Temperature contours in two planes; the front plane, which also intersects the equipment panels, gives the temperatures in both the fluid and solid elements

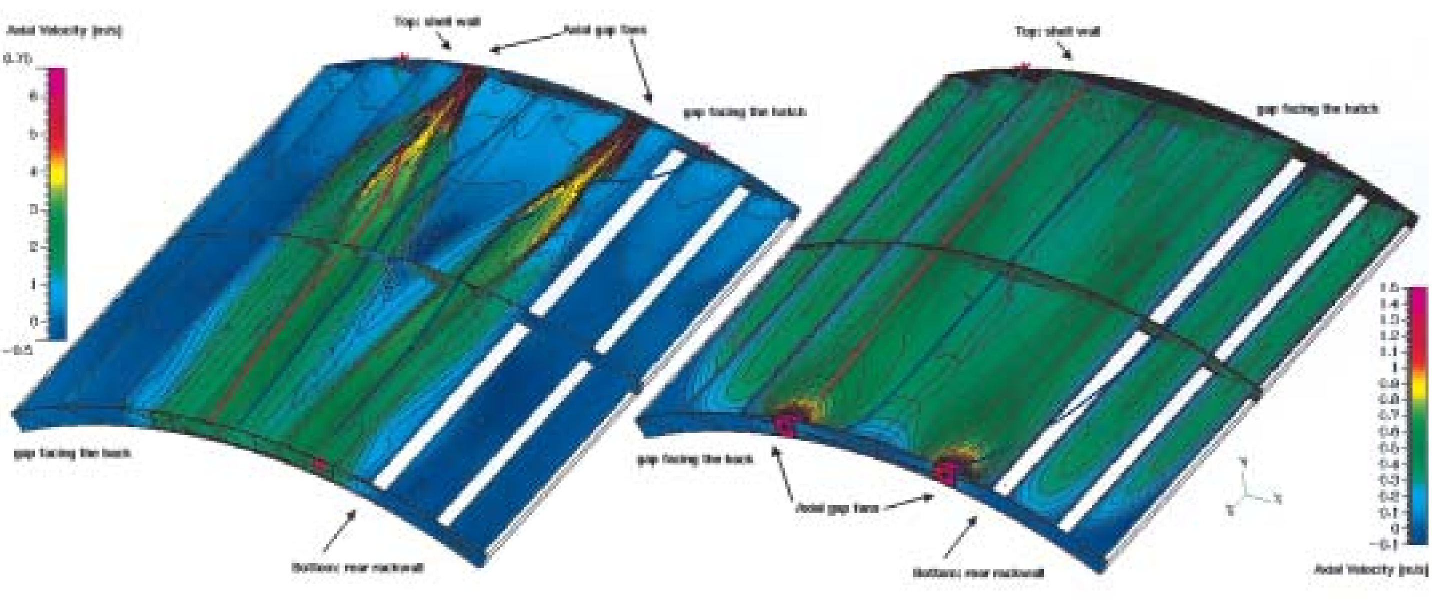
Figure 6. Flow field for a ventilated air gap with: (a) fans placed at the front end pushing air into the gap, (b) fans placed at the rear end sucking air from the gap

metabolic heat from two crew members, lights, etc. Figure 5 gives an indication of the temperature field in two planes perpendicular to the main axis. The front plane intersects the equipment panels on which the electronic boxes are mounted. One can clearly see the higher temperatures inside the panels due to the conductive heat transfer. This allows the evaluation of surface touch temperatures, which depend strongly on the convective, conductive and radiative cooling.

Within the global ventilation analysis, a specific issue investigated was the possible effect that gap ventilation behind the racks might have on the overall flow and temperature field. The idea is to take advantage of the air heat exchange with the outer ICC shell to improve the cabin air temperature. The optimal location for auxiliary fans needed to be identified to realise as uniform a flow field as possible. Different configurations were investigated, of which only two are shown here. The configuration with two axial fans pushing air into the slot only delivers two fast jets along the axes of the fans covering only a third of the cross-sectional area (Fig. 6a). This flow pattern is far from ideal and a large portion of the shell wall is unaffected. On the other hand, closing the rear gap end apart from openings for the two sucking fans allows the generation of an almost uniform flow field in the air gap (Fig. 6b).

Columbus: venting payload chambers

The International Space Station (ISS) is designed for carrying out experiments without the influence of gravity. The equipment on the ISS, however, includes many potential vibration sources that could disturb the microgravity environment. All such equipment therefore has to be designed and verified against strict specifications. The perturbations are due to the acceleration and deceleration of moving parts (pumps, fans, motors…), impact forces (opening or closing of valves), or they can be of fluid-dynamic origin. The latter happens, for instance, when a payload chamber needs to be evacuated via Columbus' vacuum and venting system. The air or other gas is released via a shut-off valve into a duct, which is connected to a venting nozzle at the port cone of Columbus. The outlet is designed to produce minimal thrust and therefore minimal microgravity disturbance. The gas plume, however, expands at high velocity once it has left the venting nozzle and impinges on the surrounding module surfaces. Figure 7 shows



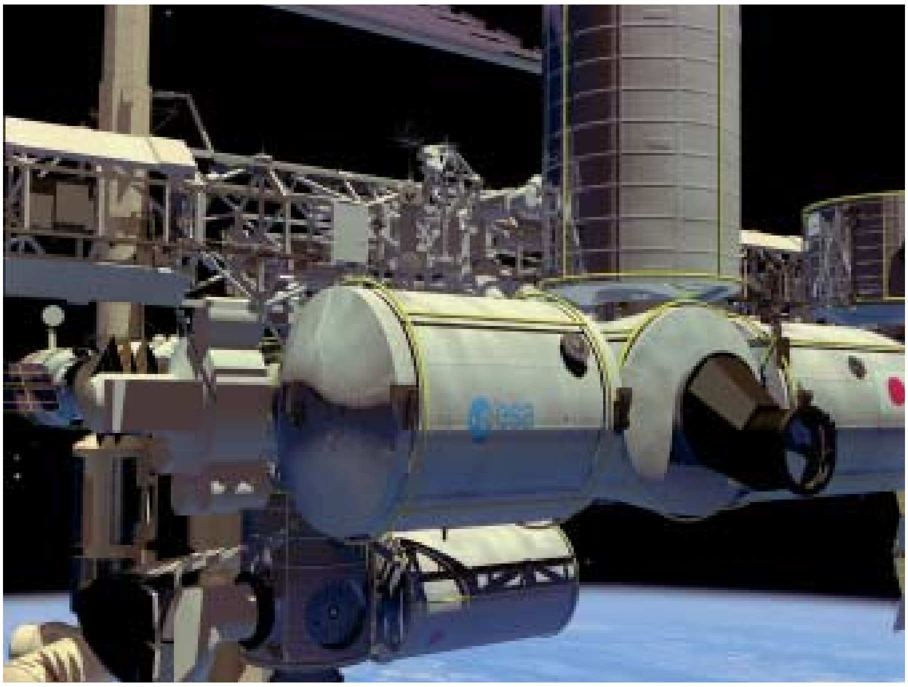


Figure 7. The European Columbus laboratory attached to Node 2, to which the Japanese laboratory (right) and the Centrifuge (top) are also docked

that the nozzle is located in a rather narrow space between Columbus, Node 2 and the Centrifuge Accommodation Module (CAM). Gas released from the venting nozzle will therefore impact several surfaces and the resulting reaction forces are not easy to predict.

The CFD tools allow calculation of the flow field and plume-impingement forces. Figure 8 shows the pressure contours on the port cone of Columbus and on the shell of Node 2. The main plume impingement is on the base plate of the venting nozzle, on a part of Node 2's surface, and on the docking adapter of the CAM. The four black lines represent the traces of the particles leaving the venting device. Figure 9 shows a more detailed pressure map in the vicinity of the venting nozzle. The results of the analysis can serve to evaluate different venting scenarios in which the gas is released in a controlled way by slowly opening the shut-off valve at the payload chamber. In this way, the microgravity disturbances can be kept to a minimum.

Conclusions

The design and evaluation of manned spacecraft habitats require an investigation of both the ventilation flow and temperature fields, driven mainly by crew comfort and safety requirements. Convection needs to be properly addressed in the prediction and analysis of the overall heat and mass transfer. Computational Fluid Dynamics has been shown to be valuable in this context, and has been successfully applied to the ATV project. Other areas of interest where the use of CFD has strong potential are, for example, heat exchangers, tuning of thermal protective systems for re-entry vehicles and combustor liners, furnaces, microgravity payloads, coolers, etc. The microgravity perturbation study for the Columbus' venting device clearly demonstrates this potential. The versatility of present CFD codes permits the simultaneous coupling of CFD analyses with other approaches, allowing a more complete analysis and optimisation to be made for any given problem.

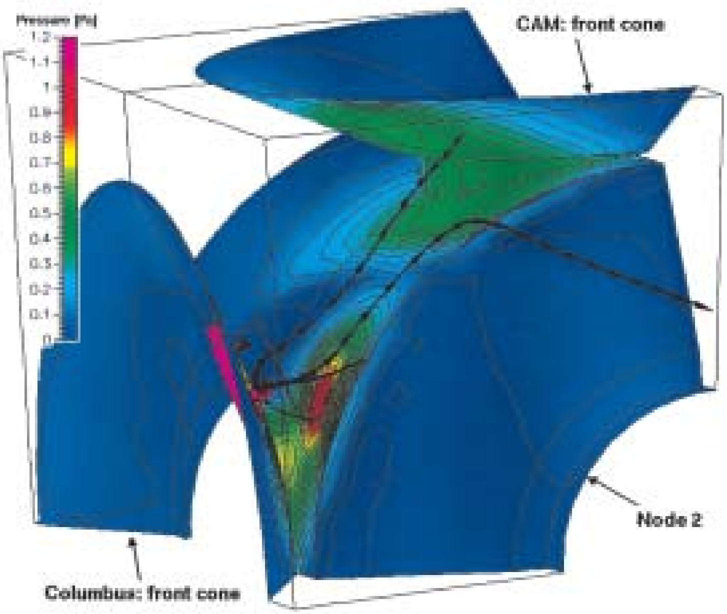


Figure 8. Pressure map for the front cone of the Centrifuge Accommodation Module and for the shell of Node 2 due to the expulsion of air into space

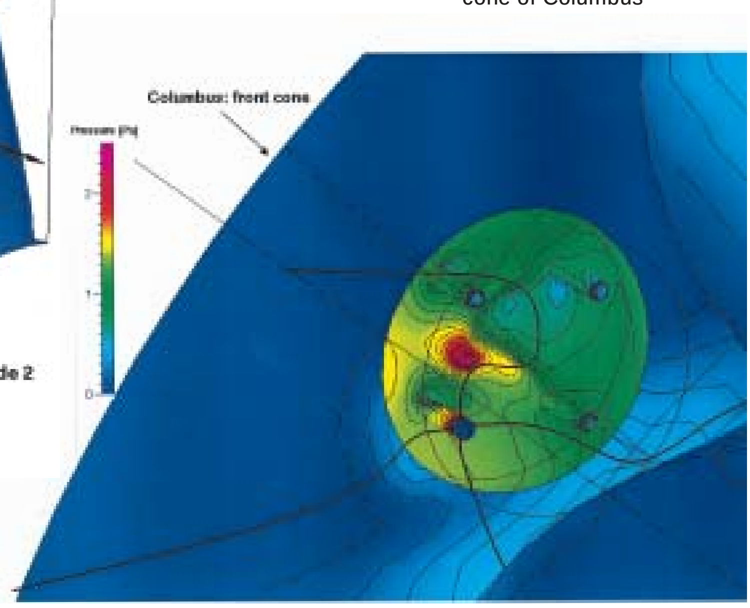


Figure 9. Pressure map showing particle traces in the vicinity of the venting device sited on the front cone of Columbus