NUMERICAL SIMULATION OF HYDRODYNAMIC LOADINGS ON REENTRY VEHICLE DURING SPLASHDOWN

Aksenov A.A.*, *, Dyadkin A.A.**, Mikhailova M. K.*,**
Schelyaev A.E.*, Simakova T.V.*, Zharckova V. V. *
*** TESIS Ltd, Moscow, Russia, ** JSV «RSC Energia» Korolev, Russia, *** JIHT of RAS**

Keywords: *CFD, numerical simulation, spacecraft, design*

Abstract

The reentry vehicle of a manned transportation spacecraft developed at RSC Energia has a segmental-conical shape. It uses a parachute-jet landing system and normally provides soft landing on the ground. In emergency situation the reentry vehicle (REV) can splash into water. In this case it is necessary to determine hydrodynamic forces and moments exerted onto the REV after contact with water as well as to study the REV dynamics after splashdown.
This paper presents the results of numerical simulation of hydrodynamic effects on the REV in the course of splashdown using CFD software FlowVision. The results are obtained by specialists of "TESIS" Ltd. and RSC "Energia".
Two regimes of splashdown are investigated: free falling, when spacecraft splashes into still water with turned off jet engines; and jet braking, when splashdown is performed with running jet engines. Pressure distribution over the REV surface and 6DOF dynamics of the REV during contact with water and submerging are presented. Air flow and water flow structures, water surface shapes and integral forces and torques acting on the REV at different time moments are presented for both splashdown cases. All this information allows engineers to analyze the transient process of spacecraft-water interaction and to select an optimal scenario of splashdown.

1 Introduction

Reentry vehicle (REV) (Figure 1) of a manned transportation spacecraft (MTSC) developed by Russian Space Corporation "Energia" performs soft landing on the earth surface with use of parachute-jet system and landing equipment [1]. An emergency splashdown of the REV on water surface is possible. In this case spacecraft designers must know the level of hydrodynamic loadings during the REV contact with water and to define the REV dynamics after splashdown. Similar problems were solved during development of manned spacecrafts "Souz"

Fig. 1. Reentry vehicle of a manned transportation spacecraft

(Soviet Union) and "Apollo" (USA). Note that for "Apollo" spacecraft splashdown is a standard situation, landing on ground is an emergency case. A huge number of experiments have been performed in the course of study of the "Apollo" splashdown which brought to light serious problems [2]. Reliable simulation methods did not exist in the times of designing spacecrafts "Souz" and "Apollo". All the results came from experiments and analytical assessments. They were limited and too approximate.

Recent advances in numerical methods, supercomputing technologies and development of verified software for computer aided engineering (CAE) allow successful using them for solving challenging problems of spacecraft interaction with water during splashdown. Numerical simulations make the safety spacecraft design more reliable.

Results of numerical simulation of reentry vehicle of Energia's MTSC splashdown is presented in this paper. All simulations are performed using CFD software FlowVision [3], recently used for solving different engineering problems involved in REV design process [4,5]. Two regimes of splashdown are investigated in this paper: free falling, when spacecraft splashdown is performed on calm water surface with turned off jet engines; and jet braking, when splashdown is performed with running jet engines. Pressure distribution over REV surface, 6DOF dynamics of REV during contact with water and submerging into the water are presented. Air flow and water flow structures, water surface shape, integral forces and torques, acting on the REV are presented for both splashdown cases at different time moments. All this information is used for detailed analysis of spacecraft-water interaction in the course of splashdown. It must help an engineer to choose an optimal splashdown scenario.

Initial data for analysis

A 3D geometrical model of the external REV shape is built in an external CAD system. It is assumed that the shape does not change in the course of splashdown. Two coordinate systems are used in calculations: associated system OXYZ and global coordinate system $O_{FV}X_{FV}Y_{FV}Z_{FV}$ - see Figure 1. The former is used for calculation of the pressure distribution over the REV surface and computation of the integral hydrodynamic forces and moments exerted onto REV, the latter is used for describing the REV dynamics. The associated coordinate system OXYZ is attached to the apparatus and moves with it. The direction of axis OX coincides with the flight direction (Fig.1). Note that in accordance with GOST 20058 2001, the longitudinal component of the force (along the OX axis) is positive if it is directed against the positive direction of OX.

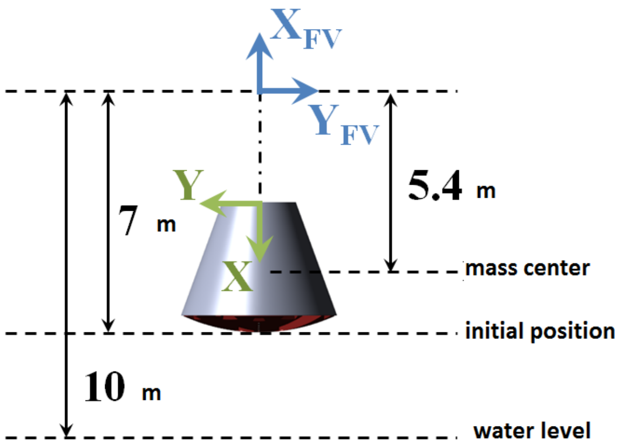

Fig. 2. Coordinate system associated with REV

The following information serves as initial data for numerical simulation of the process of REV landing:

- free drop of REV after shooting the parachute starts when the distance from the critical point of the protective screen to water surface (*H*) is equal to is 3 meters;
- the initial REV speed at this altitude is equal to 9.8 m/s
- the angular REV speed around the mass center is zero at the moment of parachute ejection;
- the water surface is calm;
- the coordinates of the mass center are as follows: x_T/L = 0,58, y_T/L = -0,02, z_T/L=0, L= 3,698 m;
- the REV mass is equal to 6,8 tons;
- the wind speed at the landing site is taken equal to 0 m/s;
- the main moments of inertia IA are: Jxx = 14600 kg*m ,Jyy = 12900 kg*m; Jxx = 12900 kg*m;

Numerical simulation method

CFD code FlowVision is used for the numerical simulation of landing REV on the water. FlowVsion uses a finite-volume approach. Simulation of REV splashdown assumes solving the problem of motion of two phases – liquid and gas. The two phases have significant difference in the densities (about 1000 times) and the values of the Mach number (in water the Mach number is almost zero, whereas the air / gas flow can be supersonic). This problem is one of the most challenging ones in computational hydrodynamics. A multi-phase VOF method is implemented in CFD code FlowVision [5]. It is used for tracking the interphase between two immiscible fluids. The amount of each continuous phase in a computational cell is characterized by the value of relative volume (VOF function). The sum of the phase volumes is equal to 1. The distribution of this function is used for reconstruction of the contact surface between the liquids. After the reconstruction the entire simulation domain is split into two sub-domains occupied by different continuous phases. The cells containing the interface boundary are split into two parts in accordance with the volume occupied by "pure" phases. Boundary conditions are specified on the interface boundary for all variables in both phases (velocity, pressure, turbulence parameters, enthalpy etc.). The equations in both phases are integrated simultaneously by implicit method. The Navier-Stokes equations are integrated using an original velocity-pressure split algorithm. This algorithm allows calculations of supersonic and incompressible flows with time steps essentially exceeding the explicit time step. The method is compatible with the technology of moving boundaries implemented in CFD code FlowVision. The method allows robust solving the transient problem of REV splashdown, interaction between the supersonic jets and water surface being taking into account. The VOF function transfer is simulated with use of the explicit method based on an advanced scheme of reconstruction of the solution inside a cell. Special attention in the VOF method, implemented in FlowVision, is paid to the modeling of those parts of the liquid, which can't be resolved by computational mesh – drops, bubbles, or a thin veil of liquid. In the traditional VOF method these formations, if they couldn't be resolved by the mesh, are simply removed from the calculation, in the present VOF method, they are accounted and handled in a special way.

All the equations are solved on a multi-processor hardware. The FlowVision algebraic solver takes into account heterogeneous parallelism of modern computers having both distributed and shared memory [6].

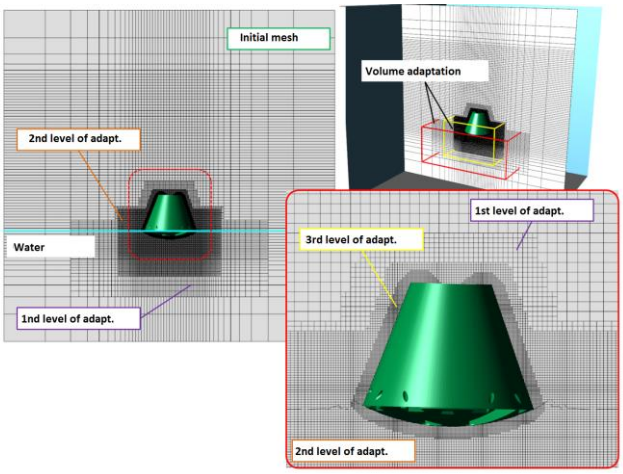

Fig. 3. Computational mesh with local dynamic adaptation

Calculation mesh

FlowVision CFD code automatically builds a computational mesh for the problem discussed. A user specifies a non-uniform initial mesh and a number of adaptation criteria. The adaptation criteria change the computational mesh in the process of calculations. Non-uniform initial mesh (see Fig. 3) is condensed in the region of the proposed trajectory of the object (in our example – linear motion of REV). The size of the initial mesh is 165 thousand cells. To obtain more accurate solutions, adaptations in volume and on REV surface (BC) are used. At the beginning of the calculation there is no need to resolve too accurately the water surface, so only the region around REV is refined. In order to minimize computer resources, the cells are merged behind REV. Applying adaptation increases the computational mesh up to 3 or 5 million cells.

Computational resources

A computer having 6 processors is used for the calculations discussed. Each processor has 4 cores. Average step of time integration is 10^{-3} seconds. For mesh containing 3.5 million cells, the wall time for one step is about 60 seconds. Simulation of splashdown during 5 seconds requires 75 hours of machine time.

Results of simulation

After free falling from a height of 3.0 meters, at moment of water touching the REV has speed about 12.26 m /s. A pitch angle θ is increasing from 0 to ~ 3 degrees because of shift of the mass center of REV from symmetry axis, As a result, at entering the water at a non-zero pitch angle plots of the pressure distribution over the REV surface are asymmetric in the plane of symmetry (Fig. 4).

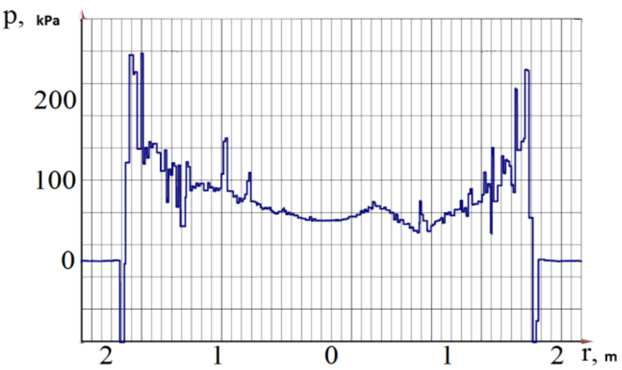

Fig. 3. Plot of relative pressure in the XY plane at time 0.3 sec, r is the distance from the center of the REV screen.

Contact with water produces sharp deceleration REV, followed by REV immersion into water. The maximum depth of the REV mass center immersion is 0.33 meters. After the first diving into the water, the REV performs several damped oscillations. Its mass center remains above the still water level in the process of this oscillatory motion. Dependence of mass center position on time is shown in Fig. 5. The position of the REV mass center is defined in the global coordinate system shown in Fig. 2.

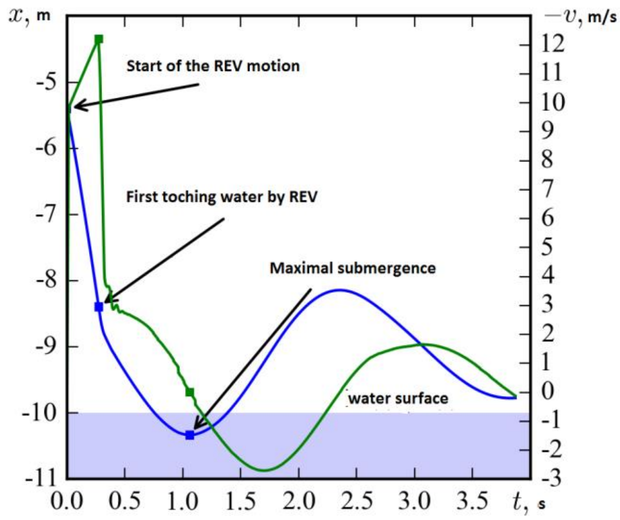

Fig. 5. Position of REV mass center and REV speed vs. time.: ■ – *vertical coordinate of mass center,* ■ – *vertical speed of REV.*

The average pressure exerted onto the REV shield is shown in Fig. 6 as a function of time. Starting from the moment of the first REV contact with water, pressure increases from 0 to 7400 кPa for a small time interval (Δt ~ 0,002 s). Further the relative pressure sharply decreases and even changes sign. The minimum pressure at the shield riches -5 kPa.

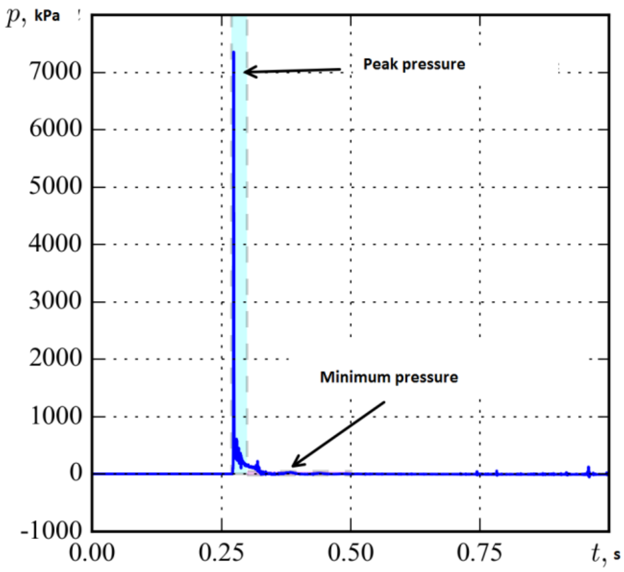

Fig. 6. Average pressure over REV shield in time.

Maximum local pressure at REV shield surface is shown in Figs. 7 and 8. The maximum local pressure is up to 250÷5500 kPa

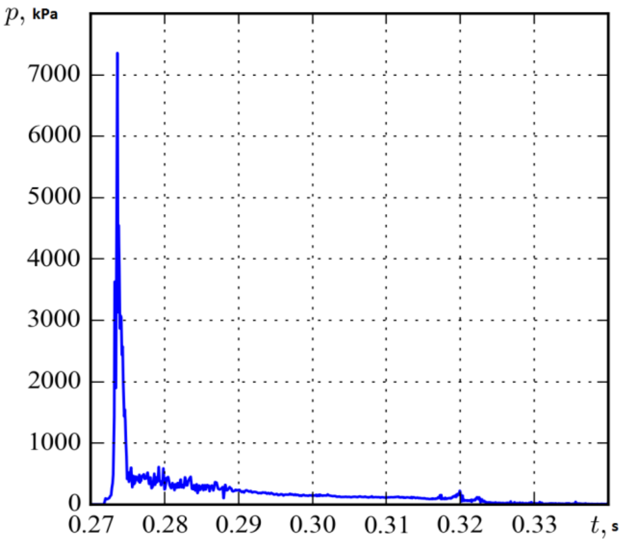

Fig. 7. Peak relative pressure on REV shield.

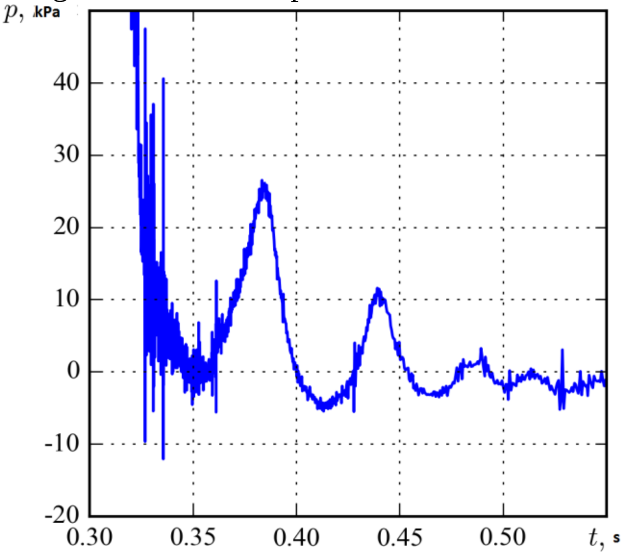

Fig. 8. Minimum expansion pressure on REV shield.

At moment of REV surfacing (t ~ 1 s) after first submerging, pressure on shield surface is decreasing and expansion pressure reaches minus 10 кPa. Aerodynamic force acting on REV reaches ~ 223 ton-force (Fig. 9).
One of the first time moments after contact with water surface for free falling landing case is shown in Figure 10.

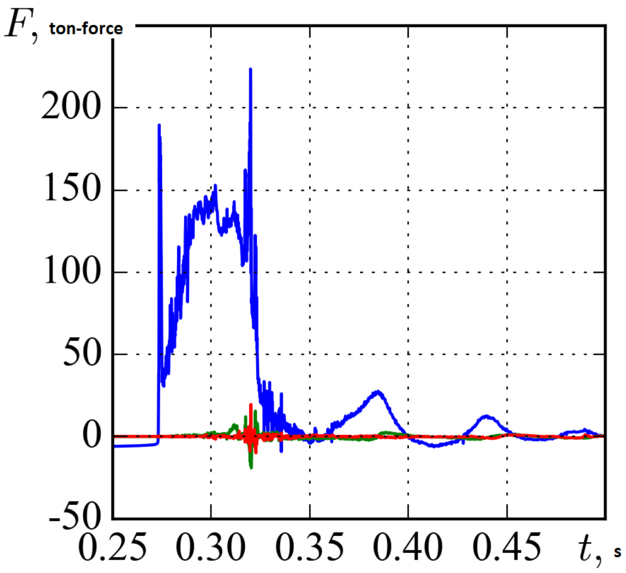

Fig. 9. Aerodynamic force acting on REV, coordinate system is in Fig. 2: ■ – *vertical*, ■ – *normal*, ■ – *transverse.*

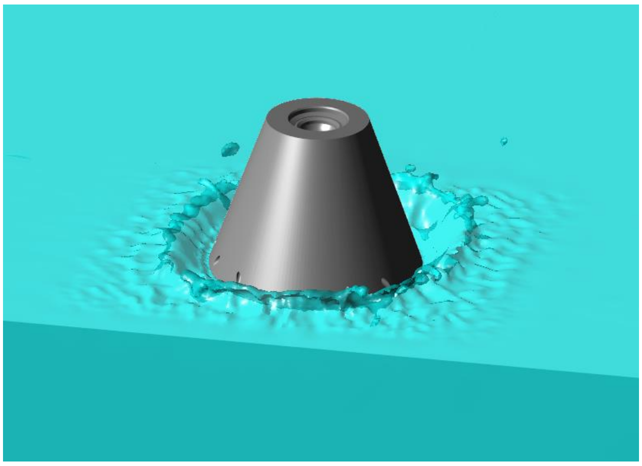

Fig. 10. Free falling case of REV splashdown. Water distribution

When jet engine is running, supersonics jets from REV nozzles "dig" out cavities in the water, generates wave going from landing area outside and create a "hump" of water directly under the REV (Fig, 12). This «hump» provides contact with water by REV at less altitude under water than for case of free falling landing. This effect is shown in Fig. 12, where is dependence of a vertical speed of REV on altitude under water. Simulation of the REV landing in both cases starts from altitude 10 m under water with the same vertical speed 10 m/s.

When we have free falling case, the speed is increased and is about 16 m/s before collision with water. REV sharply decreases the speed with high overload up to 5 m/s during collision and then continue to brake during submerging, but with smaller deceleration.

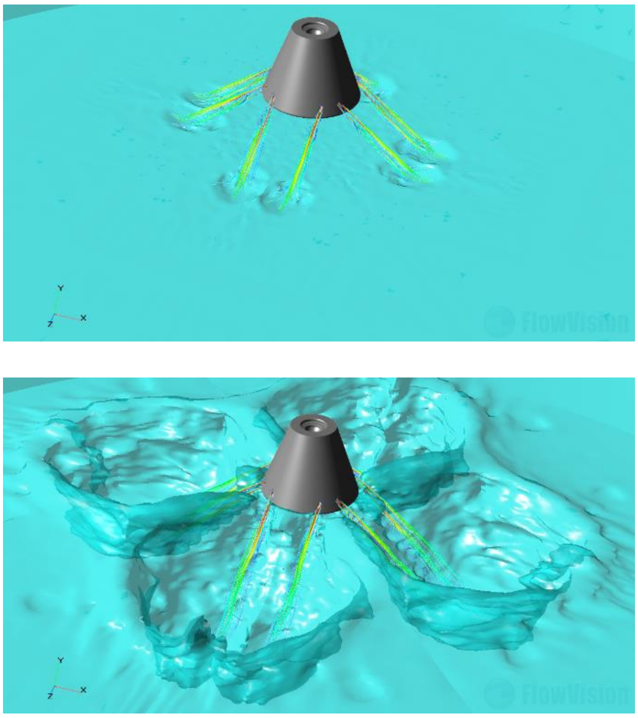

Fig. 11 Jet braking case of splashdown of reentry vehicle. Water distribution and streamlines of jets from engine.

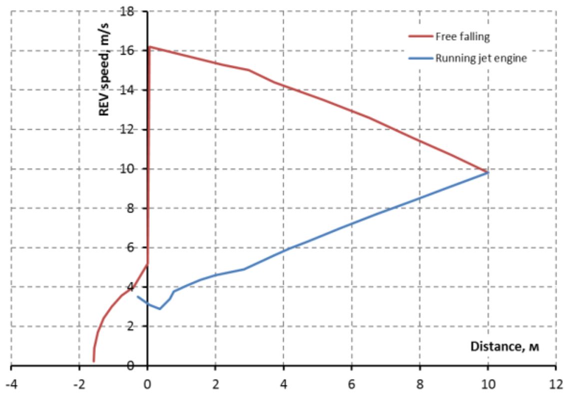

Fig. 12. Vertical speed of REV as function of altitude under water surface

For second landing case, jet system decreases the speed of the REV before contact with water, and contact occurs at altitude about 0.8 m (fracture of the speed curve at altitude 0.8 m in Fig, 12.). Jet braking system allows avoiding high overloads during splashdown. Apparently, the free falling data should be used for predicting emergency cases in the REV design.

Conclusion

The current article demonstrates that the modern numerical methods and technologies allow solving complex multi-physic problems like splashdown of spacecraft with account of interaction between running jet engines and dynamics of water surface. Advanced algebraic solver implemented in the FlowVision CFD code makes high performance calculations on multi-processor hardware one of the main methods for spacecraft design in Rocket Space Corporation “Energia”.

Investigations performed for MTSC show that a REV undergoes significant loads during splashdown. Therefore, splashdown must be considered like one of the main design cases for preserving the spacecraft structural strength and integrity of its internal equipment.

References

[1] Antonova N.P., Bryukhanov N.A., Chyotkin S.V - *Landing equipment of the new generation manned transportation spacecraft* // Kosmicheskaya technika i technologii (Космическая техника и технологии) № 4 (7)/2014, pp 21-30.

[2] Wang J.T., Lyle K.T. Simulating space capsule water landing with explicit finite element method// AIAA Paper 2007-1779.

[3] Aksenov A., Dyadkin A., Pokhilko V. - *Overcoming of barrier between CAD and CFD by modified finite volume method* // Proc. 1998 ASME Pressure Vessels and Piping Division Conf. San Diego. ASME PVP. 1998. Vol. 377-1.

[4] Aksenov A.A., Dyadkin A.A. Moskalev I.V, Petrov N.K., Simakova T.V. Computer simulation of the flow and the relative motion of the reentry vehicle and the parachute compartment hatch cover in the course of their separation during descent // Kosmicheskaya technika i technologii (Космическая техника и технологии) № 2 (9)/2015, pp 39-50.

[5] Дядькин А.А., Михайлова М.К., Сухоруков В.П., Щеляев А.Е. *Математическое моделирование обтекания возвращаемого аппарата с работающей посадочной двигательной установкой* // Труды международной научно-практической конференции «Инженерные

системы – 2011», Москва, 2011 г. – Том I. М.: РУДН, 2011 – 164 с.

Contact Author Email Address

The contact author is Andrey Aksenov, mailto: andrey@tesis.com.ru

Copyright Statement

The authors confirm that they, and/or their company or organization, hold copyright on all of the original material included in this paper. The authors also confirm that they have obtained permission, from the copyright holder of any third party material included in this paper, to publish it as part of their paper. The authors confirm that they give permission, or have obtained permission from the copyright holder of this paper, for the publication and distribution of this paper as part of the ICAS proceedings or as individual off-prints from the proceedings.