Fluid-Structure Interaction in Pilot Ejection

C. S. Chew[@], R. Lohner[*], J. D. Baum[**], H. N. Cheng[#], S. H. N. Tan[***], K. H. Lee[@]
[@] Institute of High Performance Computing, 1 Science Park Road, #01-01 Singapore Science Park II, 117528, Republic of Singapore
[*] George Mason University
[**] Science Applications International Corporation (SAIC), 1710 SAIC Drive, MS 2-6-9, McLean, Virginia 22102
[#] Air Force Research Laboratory, AFRL/HEPA Bldg 824, RM 206, 2800 Q Street Wright-Patterson AFB, OH 45433-7947, U.S.A.
[***] Defense Medical & Environmental Research Institute, DSO National Labs, 27 Medical Drive #11-00, 117510, Republic of Singapore

Abstract
The physics involving pilot ejection is looked into using a coupling of two state-of-the-art simulation software packages, FEFLO and ATB (Articulated Total Body), to obtain complete interaction between fluid and structure. This coupled approach produces more accurate results than the traditional non-coupled and non-communicating computational approaches. Studies are conducted to look into the effects on the pilot(s) during ejection sequence by wind loading and the seat ejection forces. The resulting forces acting on the pilot body are derived to infer the pilot state of health and risk assessment. This simulation will be a useful tool for evaluation and design of pilot helmet, harness, seat and ejection sequence parameters.

1. Introduction

Computational simulation is a very important component in the design cycle of flight missions and platforms. Considering the numerous factors that need to be included, pilot ejection is definitely not a straightforward task to perform. The entire phenomenon associated with vehicular ejection involves time-dependent, unsteady flows, turbulent boundary layers, potential shock waves at supersonic speeds, significant wake structure, vibrational loading, aerodynamic loading and trajectories, firing mechanism with resultant forces and thermals, etc.

Flight tests and experimental setups provide the two other alternatives to adopt in the design process. However, experimental results are largely based on controlled environments which usually cannot attain the desired demanding physical similarities with the actual flight environment. In addition, flight tests with real pilots pose a certain level of risk and danger, and come with high price tags as well. Thus computational simulation in a design cycle proves to be a very critical element in terms of efficient optimization and lowering risks involved.

In order to achieve high fidelity results with very good accuracy, it is necessary to incorporate as many elements as possible which exist within the physics of pilot ejection. These include all the abovementioned factors:

- unsteady flowfield around the aircraft with the actual high-speed flow conditions found during flight operation;
- turbulent boundary layers and separated flows, especially around cockpit and the ejected entities;
- wake structures;
- shock waves (wave-wave, wave-body interactions);
- kinematic and dynamic conditions of the ejection sequence (canopy, seat, harness, belt, thrusters, etc.);
- body trajectories and loading forces;
- actual physiological responses (limbs and joints) to the forces and moments exerted by the resulting windblast.

To simulate using a single computational fluid dynamics (CFD) code will be insufficient to accurately derive the pilot body motion and response. Even by incorporating a rigid body model will one only be able to obtain a gross force distribution and resultant trajectory based on summation of the forces and moments around its centre of gravity. This kind of results will only serve at a low level of fidelity. In order to look at the exact body behaviour, a solid mechanics simulator needs to be included to solve for the articulating motions. The resulting motions will also in turn affect the overall flowfield around the body. Thus a fluid-structure coupling is required to obtain this kind of interaction.

This study was initiated in order to observe the extent of human body injury during ejection. Other than the force loading being experienced on the pilot's body, the amount of twisting around the neck and spinal area is also a primary concern. It was then determined that a CFD code be coupled with an articulating body-segment modeller. As a result, Articulated Total Body (ATB for short) was chosen for its matching capabilities, together with its working CFD partner, a robust and highly efficient code, FEFLO.

Before delving into the computational methodology, the next section provides a brief description of pilot ejection.

2. Pilot ejection

2.1 Pilot and seat

Vehicular ejection provides an escape route for the occupants of the vehicle. In order to ensure safety during escape, the ejection system must be carefully deliberated. For pilot ejection, the most basic components are the seat and pilot.

The seat ejection, presumably carrying the pilot, is a two-step process: an explosive catapult to remove the seat, and a rocket thruster to effectively separate the seat from the aircraft. First, the seat speed must be high enough to give reasonable separation from the aircraft in a very short span of time. However, the seat acceleration must not be too high to cause unnecessarily high G-forces acting on the pilot. The ejection time must be short to ensure fast clearance, but long to enable the entire sequence of events to take place. Therefore a delicate balance must be struck between ejection time, speed, acceleration and clearance distance.

Figure 1 shows the example of two different types of aircraft and the required G forces and ejection speeds. Due to the long cockpit-to-tail distance, the time to tail of the B-1 is much longer than the F-4 jet. As a result, less speed can be allowed with a lower G force acting on the pilot.

airspeed (knots)	B-1 time to tail	speed ft/sec	G force	F-4 time to tail	speed ft/sec	G force
100	0.651	24	3.0	0.240	29	3.7
200	0.325	48	6.0	0.120	59	7.3
300	0.217	72	9.0	0.080	88	11.0
400	0.163	96	12.0	0.060	117	14.7
500	0.130	120	15.0	0.048	147	18.4

Figure 1. Examples of time-to-tail, speed, G force on B-1 and F-4.
Source: http://www.showcase.netins.net/web/herker/ejection/physics.html

Another type of test that is commonly conducted is the sled test. The forces experienced by the pilot (or mannikin) will differ from the actual environment experienced by the in-flight conditions. Most importantly, the sled is made to accelerate to very high speeds (~1Ma) in very short times, causing G forces of around 30. The air conditions are at near to sea level values and ground effects come into play as well, in terms of the formation of turbulent boundary layers.

Apart from the kinematics, the forces required to thrust the pilot and seat are basically determined by the mass of the seat, pilot and attached equipment. Typical figures are shown in Figure 2.

item	mass (5%)	mass (50%)	mass (95%)
seat + survival pack + parachute	253 pounds	253 pounds	253 pounds
pilot personal equipment	40 pounds	40 pounds	40 pounds
pilot mass	103 pounds	154 pounds	205 pounds
total	396 pounds	447 pounds	498 pounds

Figure 2. Typical masses of pilot, seat and equipment.
Source: http://www.showcase.netins.net/web/herker/ejection/physics.html

2.2 Canopy & ejection sequence

Apart from the attention being paid to the windblast on the pilot, the other major safety criteria comes from possible collision, particularly with the aircraft and canopy (if it has not been broken yet).

A canopy is usually equipped with small rocket thrusters on the forward lip to push it up and behind into the slipstream, away from the anticipated pilot ejection path. For a single-pilot aircraft, this is straightforward and easier to design. If the aircraft consists of multiple passengers, or if the craft is a vertical take-off and landing (VTOL) type, additional factors come into play. The alternate solutions are then to either arrange a canopy breaker at the top of the pilot seat, or line the canopy with shape charge, so that explosion will take place at the very beginning of the ejection sequence to break the canopy and enable the pilot(s) to eject safely.

Different aircraft seats have different computer-programmed ejection sequences. Three examples are the Martin Baker MK15, the McDonnell Douglas ACES II and Universal Propulsion S4S. The ejection sequence consists of the key events as listed below:

- Ejection initiation
- Strapping pilots to seats (restraint systems)
- Airspeed and altitude conditions measurements for parachute deployment
- Aft canopy actuator
- Aft seat ejects
- Aft-to-front delay time
- Forward canopy actuator
- Forward seat ejects
- Parachute deployment

The total time is typically <2 seconds for a single seat ejection. In order to prevent collision, the thrusters also employ varying vector thrusts on the seat and canopy. An even more detailed sequence right up to canopy release is listed as such:

- Thrusters push canopy aft by ~4cm (~8kN force each for ~100ms)
- Canopy shifted up by few ~2cm
- Port-side thrusters activated first (~20° to vertical normal)
- Starboard side thrusters activated next (~20° to vertical normal)
- Canopy gets to rotate until about 45 degrees about the aft hinge, then released
  (time to release ≈ 120ms; time to rotate canopy ≈ 60ms)

2.3 The sled test

Typically, a straight-lined sled track is built on an open area, and the forebody of an aircraft is placed at the beginning of the sled train. Its rear is equipped with a number of rocket sleds, with each rocket sled release (starting from the furthest rocket) representing a stage of acceleration. For example, in Figure 3, there are three rockets sleds. The rear two sleds will be released somewhere along the track, triggered by screen boxes. These screen boxes hold electric contact knives to activate release and firing up of the rockets.

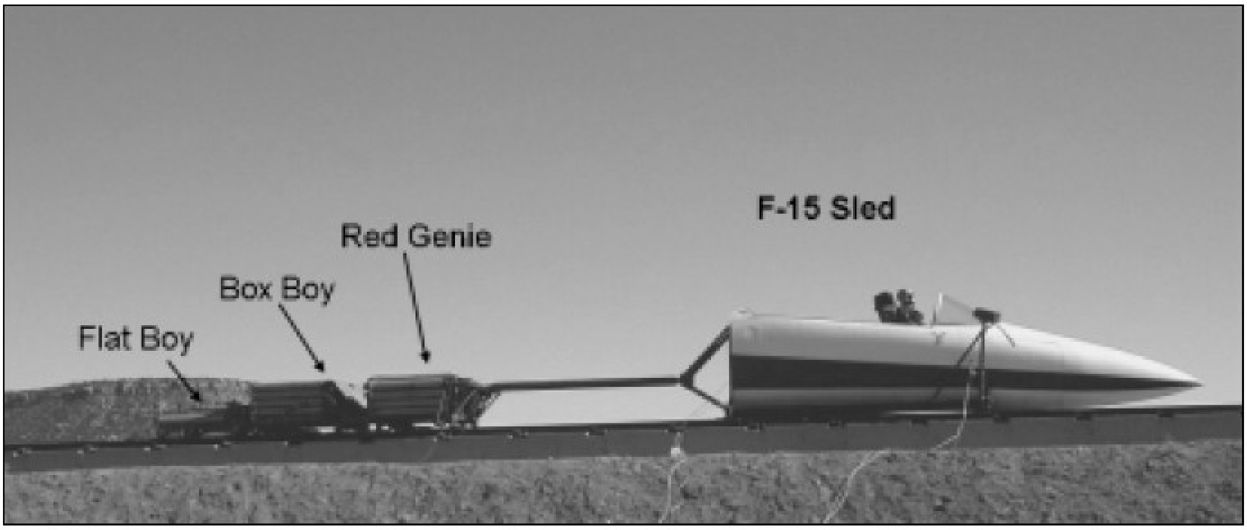

Figure 3. Three rocket sleds with forebody setup. “Fat Boy” gets released first, followed by “Box Boy”. “Red Genie” provides the final boost and is attached to the forebody throughout the test.

As mentioned previously, the sled test, though expensive, is a much better alternative to sending pilots up flying and performing actual ejection, for obvious safety and financial reasons. Mannikins can be deployed in this setup and they carry tracking and measurement equipment. The acceleration from zero to transonic happens in a very short time, and G forces of around 30 are usually experienced. In addition, this type of setup will only enable testing of single-direction wind loading. At the expense of such space, time, equipment and effort, the limitations of the controlled environment sometimes seem unacceptable. This further reinforces the importance of computational simulation, with its fast turnaround time, effective design optimization, and a much wider range of allowable scenarios and test inputs.

3. FSI & Coupling methodology

3.1 Fluid-structure interaction

In order to effectively model the physics of pilot ejection, a traditional CFD or CSD (computational solid dynamics) approach is insufficient to investigate the solid mechanics behaviour and its reciprocating effects on the flowfield. With the advancement in computer technology, the path to undertake will be that of coupled methodology in a preferrably parallel environment.

A standard approach is by transferring the displacement vectors at the solid boundaries to the CFD solver, which it will treat as the boundary conditions for the fluid domain. The flowfield is then solved to obtain the boundary pressure and stress values. These sets of information are then passed to the CSD solver to be treated as its solid boundary conditions in terms of forces. The solid governing equations can then be solved to obtain a next set of boundary displacements, which will then be passed back to the fluid solver. Thus the cycle continues to obtain a more realistic and interactive field solution.

To date, there are a number of ways to implement the coupled methodology. The strictest approach is to adopt a single mesh for both the solid and fluid domains. The governing equations are different for each phase, but solved altogether nonetheless. This is the monolithic approach. Another strategy is to enable the solid mesh and fluid mesh to coincide at the interface, to obtain a partitioned approach. However, in most of the occasions, the fluid solver and solid solver are written separately. In this case, the most feasible way is to adopt loose coupling. The meshes do not have exact matching, but interpolation can be performed to exchange information. A communicator programme is sometimes written to execute the interpolation, array manipulation and transfer. Also, due to the difference in time step sizes, attention needs to be paid to the sequencing and advancement of the time-integrated solutions to ensure stability of the solution.

3.2 FEFLO & ATB

The present study employs two codes: FEFLO and ATB. FEFLO is a finite element CFD code which is able to solve a wide variety of flow problems. Because of the requirement to investigate the human body segment and joint behaviour, ATB is chosen as the biomechanical solver.

FEFLO is a robust, state-of-the-art, continuum-based CFD solver being developed by SAIC and George Mason University. It handles many forms of flow physics like aerodynamics, hemodynamics, blast and fragmentation, acoustics, etc. In terms of user interface, it has its own pre and post processors, and easily parallelises jobs in all sorts of machine platforms. The modules and their functions include:

FEFLO PRE and FECAD	Pre-processors. FEFLO PRE is a Windows version developed by SAIC.
FEFLO	CFD solver.
FEMAP	Handles FSI problems by managing FEFLO and a CSD solver.
FE CSD	CSD solver.
FE EIGEN	An alternate form of CSD solver which looks at the eigen modes of the solid structures.

Because of the unsteady nature of the solid-fluid interface, mesh movement and/or remeshing are necessary. The efficient way to handle moving boundaries in CFD is via an arbitrary Lagrangian-Eulerian (ALE) formulation of the governing equations. For a computational element moving at a velocity W,

$$W = \{u_E, \quad v_E, \quad w_E\} \tag{1}$$

The fluid governing equations in ALE formulation can be written in the form as such:

$$\begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e \end{Bmatrix}_{,t} + \begin{Bmatrix} (u^i - W^i)\rho \\ (u^i - W^i)\rho u + p \\ (u^i - W^i)\rho v \\ (u^i - W^i)\rho w \\ (u^i - W^i)\rho e + u^i p \end{Bmatrix}_{,i} = -\nabla \cdot W \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e \end{Bmatrix} \tag{2}$$

where ρ is the fluid density, p is pressure, u, v, w, e are the three velocity components and internal energy respectively. When W is equivalent to the boundary velocity, the equations is of Lagrangian nature. When $W = 0$, the equations become Eulerian in nature. If a set of elements that are attached to the solid body move in the exact motion as the solid body, the arbitrary velocities W can easily be found, and the governing equations solved in ALE form.

The **Articulated Total Body (ATB)** Model is used by the Air Force Research Laboratory (AFRL) and various other organizations, companies, and educational institutions for predicting gross human body response in various dynamic environments, especially automobile crash and aircraft ejection with windblast exposure.

The ATB Model originated from the Crash Victim Simulation (CVS) Program. Aerodynamic force application and a harness belt capability were added to the CVS Program, and the resulting program became known as the ATB Model. In 1980, several modifications to the ATB Model were made, combining it with the then-current 3-D CVS program to form the ATB-II Model. The next version, ATB-III, was generated to model the body response to windblast for AFRL. ATB-IV was released in 1988 with a number of additional efforts being made to improve various aspects of the ATB-III Model, with emphasis on its capability to simulate aircraft ejection with windblast exposure as well as complex automobile accidents.

The ATB-V Model introduces three new simulation tools: water force simulation, robotic simulation, and deformable segments. A major change has been made to the data arrays to increase the maximum number of segments, planes, and contact definitions. A new type of structured ASCII graphics data output file has also been designed.

Figure 4 illustrates ATB at work. Each body segment can be modeled along with specifications at the joints. From Figure 4, the resulting motions are well compared with experimental footage. In addition, Figure 5 shows the many number of ways to model the joint types. In conclusion, ATB can be used to accurately model the human body biomechanical behaviour.

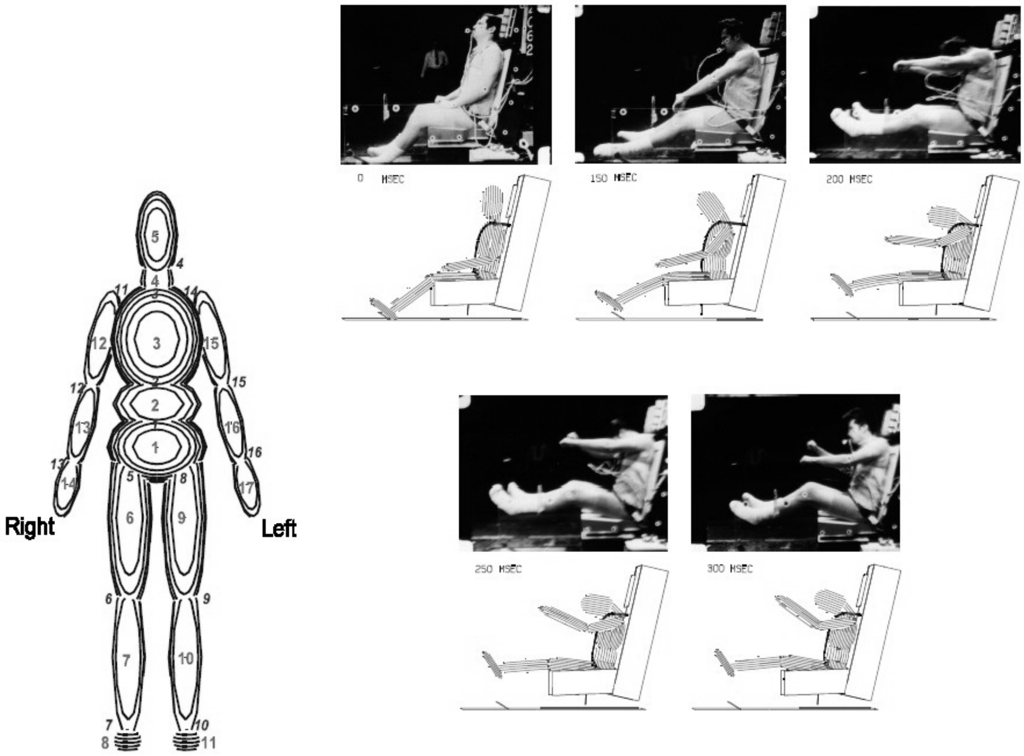

Figure 4. Left: typical segment and joint setup in ATB programme. Right: ATB simulation results compared to actual test footage.

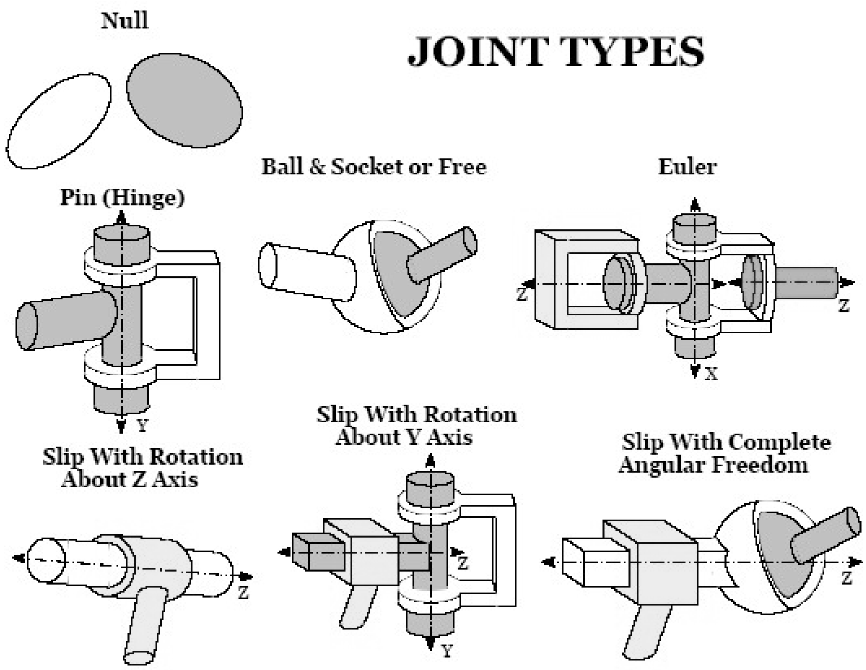

Figure 5. Available types of joint configurations between segments.

FEFLO has engaged in pilot ejection simulations before (Refs. [1] & [2]). However, the seat and pilots were primarily modelled as solid rigid bodies with no biomechanical responses (Figure 6, left). With the inclusion of ATB, a more realistic simulation can be achieved, and more in-depth knowledge in pilot injury and prevention can thus be gained.

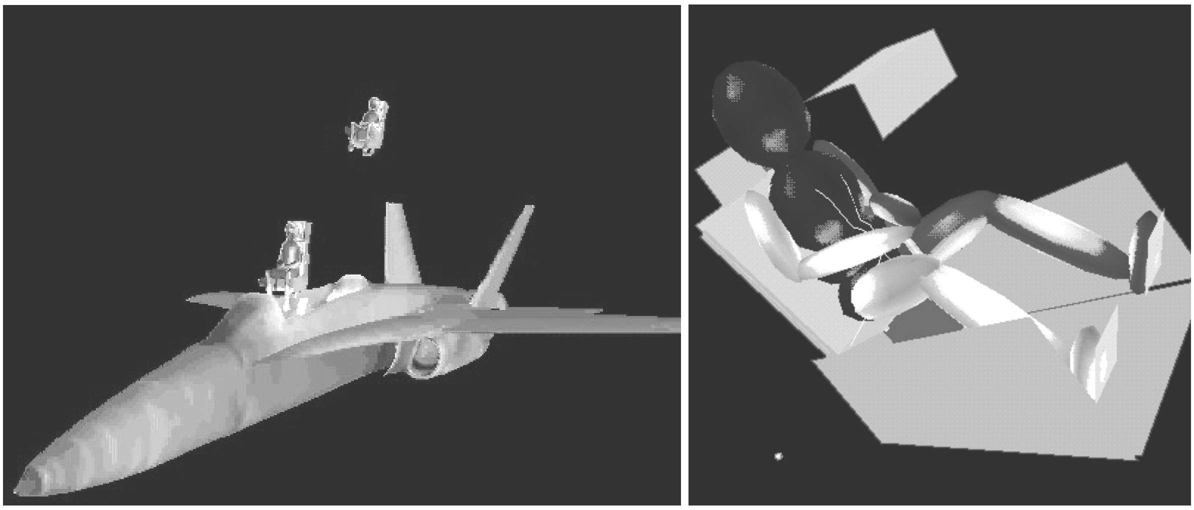

Figure 6: Pilot ejection simulation of F-18 fighter craft (left) and ATB biomechanics software (right).

3.3 Problem setup

The primary concern at this stage of study is the injury assessment of the pilot in an F-16 aircraft (forebody) during a sled test execution. The basic domain should be a long stretch of 'runway' in an open field environment. The rocket sleds need not be included, but the required accelerations and timing sequence must be implemented onto the aircraft in the form of prescribed motion.

The actual sled is accelerated from zero to transonic speeds, and that poses a problem to the CFD solver in terms of the nature of the governing equations. At low speeds, an incompressible Navier-Stokes solver is typically preferred. However, as the relative flow gets past the subsonic regime, the compressibility effect gets more and more significant, up to the formation of shocks at sonic regime. The Euler equations with upwind treatments are then preferred to obtain accurately the pressure jumps and shock front locations. This switch in solver is not commonly practised. Thus one way is to employ the integration of Euler equations throughout, implying that the initial low-speed stage be aborted altogether, and the sled starts with a significantly high velocity and gets ramped up to about 600 KEAS (knots equivalent air speed; ~ Mach 1).

Due to the long fluid domain, the element mesh population will be very large. The use of mesh movement and remeshing across the entire domain may not be a good strategy, considering the high overheads involved. Thus one potential solution is to attach overlapping submeshes onto the moving bodies. The submesh around the sled will follow the sled, only performing interpolation between the mesh interfaces to exchange flow information between the submesh and the main domain mesh.

Other kinematic and dynamic considerations are: seat and pilot mass, centroids, moments of inertia, joints and segments, helmet, harness, seat straps, ejection sequence timing, thruster vector and force history. All features involving the pilot and seat configuration will be amply handled by ATB, including the helmet, harness and straps. The ejection sequence and thrusters are modelled by FEFLO.

As mentioned, FEFLO has been used before to model pilot ejection, with the treatment of ejecting objects as solid rigid bodies. In Refs. [1] & [2], the compressible Euler equations were solved. The entire aircraft was immersed within a high speed flow field under a single fluid domain at high altitude. Adaptive unstructured mesh refinement with moving mesh ALE formulation was adopted. For the study involving canopy release, the exact thrust data and kinematic conditions were imposed. From Figures 7 and 8, the predicted canopy trajectories and motion were in good agreement with known experimental results. Based on these existing results, the full scale coupling work can be carried out with the knowledge of good validity in the benchmark work obtained by FEFLO.

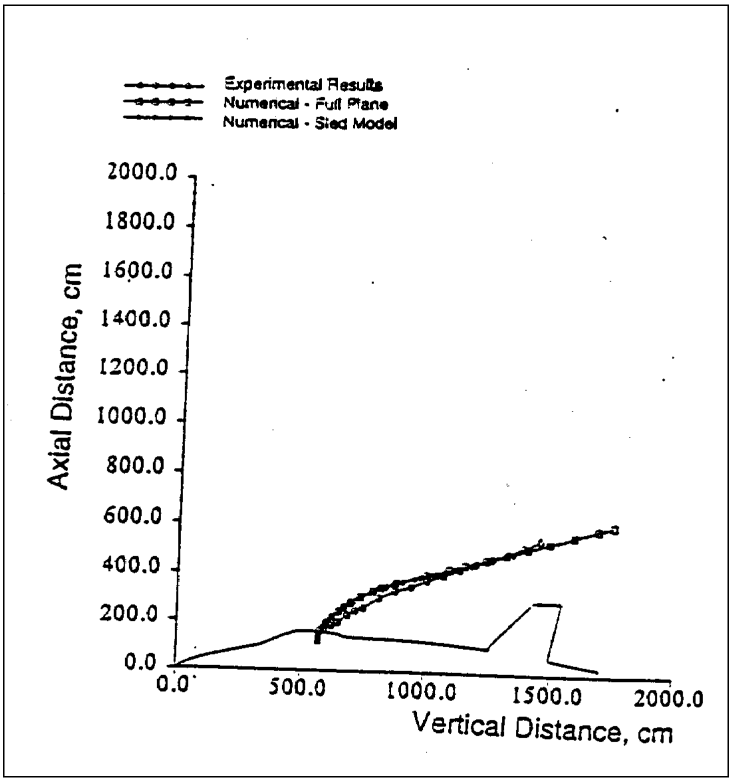

Figure 7. Ref. [1], canopy trajectory from an F/A18-C/D fighter.

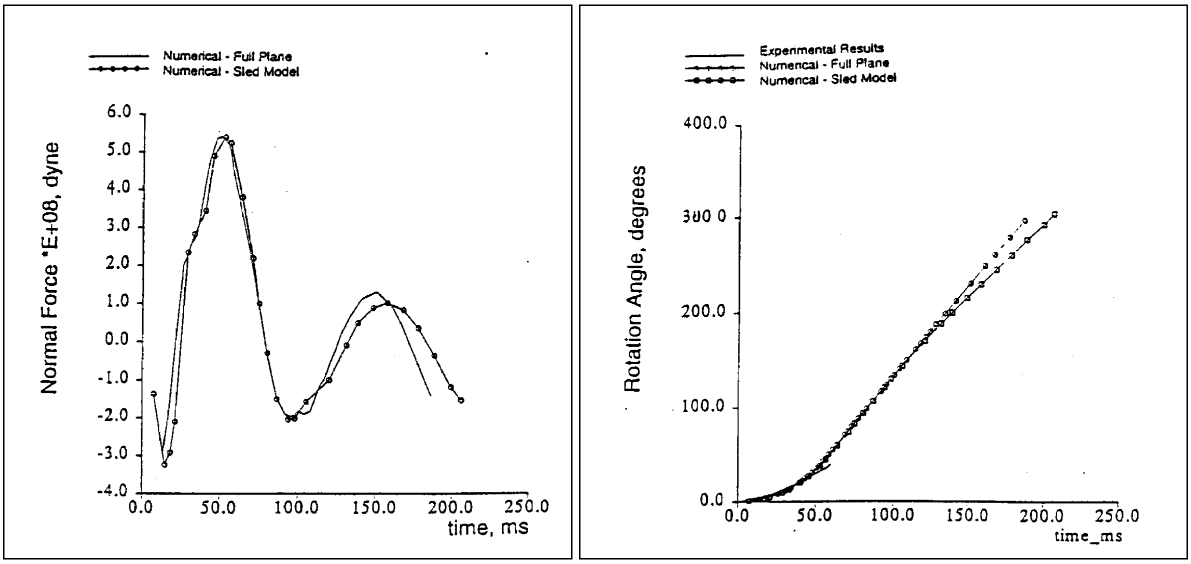

Figure 8. Ref. [1], ejection from an F/A18-C/D fighter. Left: evolution of normal force on canopy. Right: evolution of canopy rotation angle.

4. Preliminary results & future scope

A preliminary run was done by solving steady state flow (Mach ~0.7) past the F-16 aircraft on the ground using FEFLO. The canopy was not included in the computation. From Figure 9, it can be seen that ground effects will come into play when the plane runs along the path. Figures 10 and 11 indicate the type of flow speed and pressure forces acting on the plane and pilot.

It can be seen that the pilot helmet is experiencing high pressure forces. From Figure 10, a wake region of low Mach speeds was observed behind the pilot. This produces a high shear region which will definitely affect the overall interaction once the ejection sequence starts. Also, from Figure 11, a good force distribution on the pilot and helmet can be captured, enabling ATB to subsequently accurately produce realistic responses from the biomechanical model.

The next stage will involve the coupling process between FEFLO and ATB. The aircraft will be prescribed to move in a straight direction, emulating the sled track. The ejection sequence and actuator forces will be implemented onto the aircraft and seat. What remains to be seen will be the biomechanical responses produced by the ATB model, and subsequent works will be carried out in injury assessment and reduction through processes like helmet design.

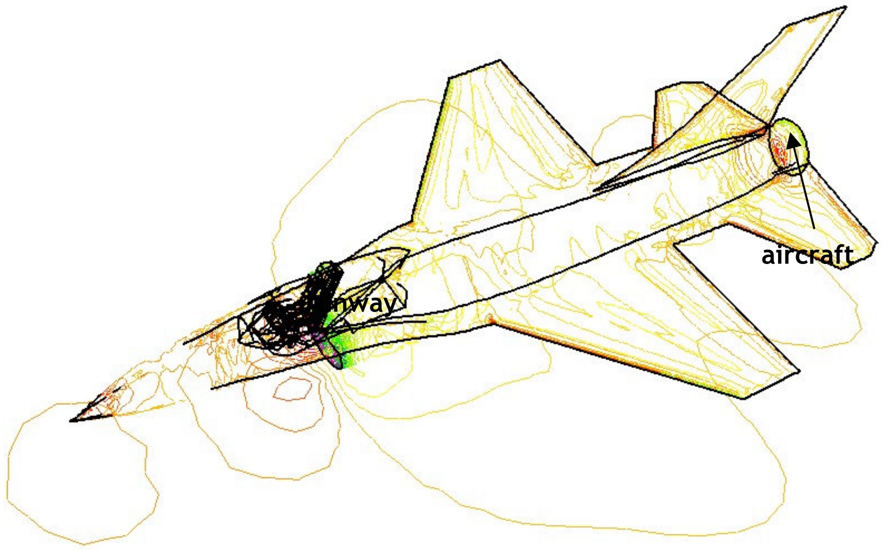

Figure 9. Pressure contours, showing ground effects from a steady-state run of an F-16 aircraft.

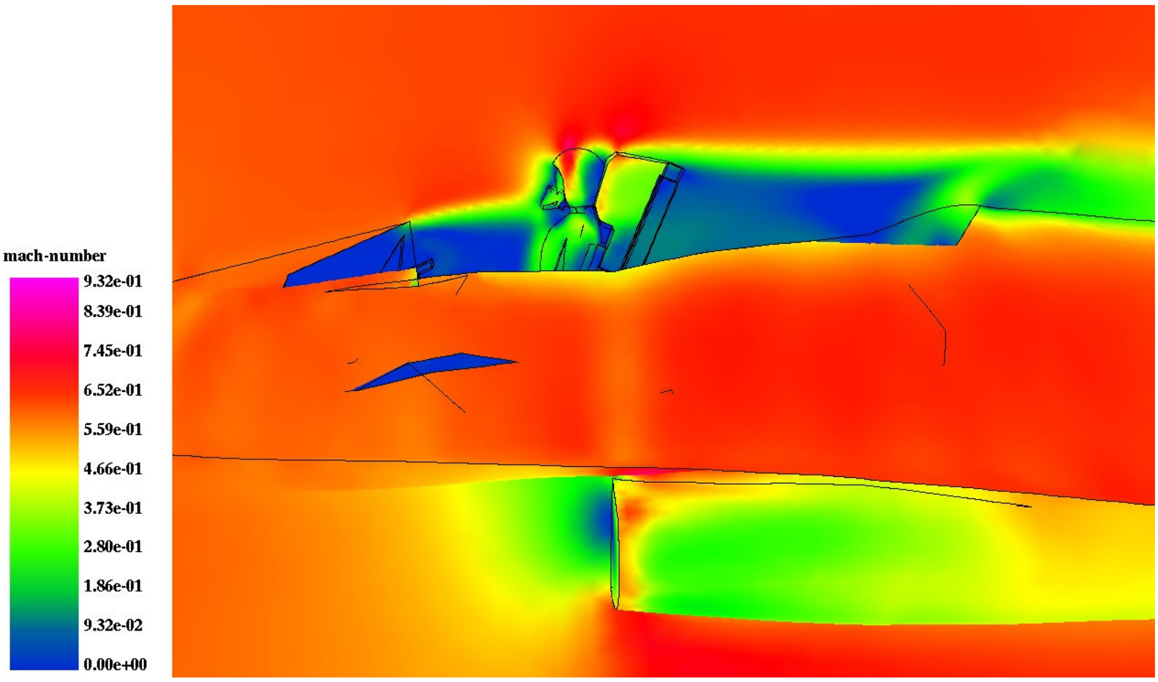

Figure 10. High Mach number experienced at the pilot head area exposed to windblast.

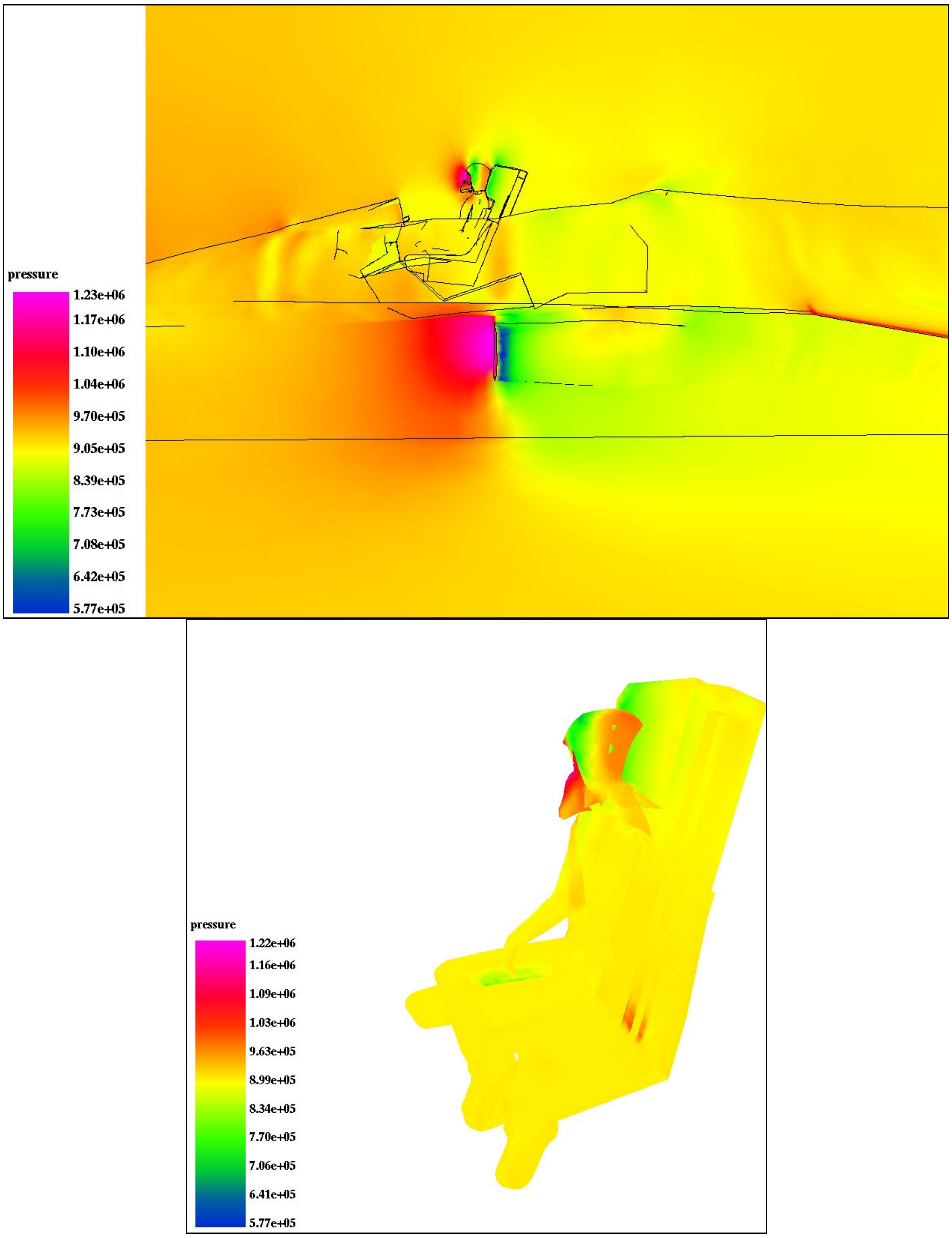

Figure 11. Above: high pressure forces exerted on the pilot head area. Below: force distribution across the helmet area, captured by FEFLO.

References

[1] Baum J.D., Lohner R., Marquette T.J. and Luo H.. Numerical simulation of aircraft canopy trajectory. *AIAA 97-1885*, *28th AIAA Fluid Dynamics Conference, 4th AIAA Shear Flow Control Conference*, 1997.
[2] Baum J.D. and Lohner R. Numerical simulation of pilot/seat ejection from an F-16. *AIAA-1993-783*, 1993.
[3] Cheng Z.Q., Rizer A.L., Pellettiere J.A. and Pilkey W.D. Limiting performance analysis of biomechanical systems for optimal injury control – Part 2: Applications. *IJCrash*, 10 No. 6: 579-587, 2005.
[4] Cebral J.R. and Lohner R. On the loose coupling of implicit time-marching codes. *AIAA-05-1093*, 2005.

This page has been purposedly left blank