18th Australasian Fluid Mechanics Conference
Launceston, Australia
3-7 December 2012

Toward the full CFD Simulation of Expansion Tubes

J. Sancho[1], D.E. Gildfind[1], P.A[1]. Jacobs, R.G. Morgan[1]

[1]Centre for Hypersonics
The University of Queensland, St Lucia, Queensland, 4072, Australia

Abstract

In order to perform ground testing in expansion tubes, it is highly desirable to characterize the facility by CFD, since the flow processing is very complex. Nevertheless 2D o 3D modelling requires tackling several questions that remains opened. The main challenge is to include piston dynamics, but for that, launcher station pressure loss must be accurately simulated. This paper explores conceptual CFD models aiming to demonstrate that these effects can be included accurately. Results show that there are significant pressure losses at the launcher brought by the supersonic flow jets found after the slots. The aim is to clear the path towards the development of a full facility model, including the long time-scale piston motion couped to the gas dynamics at the whole machine.

Introduction

The Center for Hypersonics at The University of Queensland UQ is currently performing extensive research on *Scramjet* technologies. In order to generate cost feasible test conditions for fundamental engineering and design development, ground testing is required. The only facilities known to produce conditions of the order of several gigapascals of total pressure which are required for the access-to-space upper *Mach* 10-15 envisaged trajectory [3], are expansion tubes, such as X2 and X3 facilities at UQ.

Understanding and measuring flow features at such extreme conditions is challenging. To help to characterize the flow, computational fluid dynamics (CFD) is employed to complement test data. An expansion tube uses several stages to accelerate the gas. Typically, includes a free-piston compression of the driver gas, feeding a shock tube, followed by an unsteady expansion tube, and a steady expansion nozzle towards the test section, where the model is located. This process generates rich flow features, including fluid-rigid body interaction (piston dynamics), shocks and expansion waves, contact discontinuities, multi species boundary layers, and several flow instabilities such as *Rayleigh–Taylor* and *Richtmyer–Meshkov* instabilities.

Although some of these features have successfully been simulated by CFD in the past (see for example [5]), the piston compression phase has not been widely explored beyond 1D analysis, mainly because of the difficulties in including piston dynamics (fluid-solid motion interaction). *Scramjet* conditions have been partly simulated with CFD for X2 [5], and X3 [8], using 1D models [6] and hybrid 1D/*2D* axisymmetric [7].

L1D [6] is a 1D lagrangian code developed at UQ by P.A. Jacobs that can model the entire facility, including piston dynamics. Nevertheless typical 2D/3D flow features as area changes or pressure losses have to be modelled with constants to be determined empirically. This is particularly important in the determination of the free piston launcher mechanism pressure losses which affect the performance of the entire free piston compression process.

Eilmer3 [7] 2D axisymmetric and 3D models may offer a way forward to simulate these phenomena, but piston dynamics has not been implemented yet. This fact limits the capability of modelling the compressed primary driver gas flow, since it has to be provided as a boundary condition for the rest of the simulation either using experimental analytically derived or 1D calculated values. This modelling issue is of paramount importance since it was speculated [10] that the origin of the noise test gas measured in the test section for low enthalpy conditions may be related to the primary diaphragm rupture process and the primary driver area change, and flow features derived from piston compression dynamics [5].

This paper explores solutions to these remaining questions. First considers whether the launcher pressure losses measured experimentally can be predicted using CFD simulations. Secondly, if piston dynamics can be included in an axisymmetric model in order to simulate the driver gas flow.

If these questions can be addressed, a complete 2D (or 3D simulation) of the facility would be feasible, leading to a truly complete full facility simulation.

X2 Launcher pressure losses.

One of the most important parameters leading to driver performance reduction is the launcher station head pressure loss. At the very beginning of the test sequence the piston is accelerated by the pressure differential between the front and rear face. The rear face of the piston is fed by the reservoir, and the front communicates with the driver tube. It is desired that the area of the rear is accelerated by a nearly isentropic expansion reservoir gas to maximize the performance.

Nevertheless, the facility needs a launcher mechanism that holds the piston in position at the beginning of the compression tube, while the adequate gas fill conditions are reached. When the piston is released, the air flow passes through the launcher. This process inherently creates a pressure loss, reducing the effective reservoir pressure at the back of the piston. Assuming a nearly incompressible flow process at the launcher, the loss is expected to be proportional to the local dynamic pressure. This is the model used in L1D, and the proportionality factor must be supplied.

From [4] it is observed that with a good experimentally determined loss coefficient, L1D is capable of predicting with reasonable accuracy the piston compression process. Nevertheless the experimental determination of the launcher head pressure loss factor is arduous, since must be determined for every condition. Also, experimental values obtained were much higher than expected [5], and therefore it was suspected that the formulation was not capturing the phenomenon accurately.

In addition to that, the model does not shed any light on the pressure loss flow process, therefore there is no much room for its improvement. In order to better understand the flow process at the launcher station, and with the aim to estimate the pressure loss factor without the need for blanked off testing, some CFD models were therefore attempted. CFD calculation of localized head pressure losses is challenging because the tridimensionality of the flow and its *Reynolds* and *Mach* numbers dependence nature.

CFD launcher model

An X2 CAD model was imported in *igs* format to ICEM commercial grid generation suit, figure 1. Since the standard procedure to obtain launcher pressure loss factor is performed using a blanked off test (closing the primary diaphragm with a plate), only the reservoir, launcher, piston and driver tube are used.

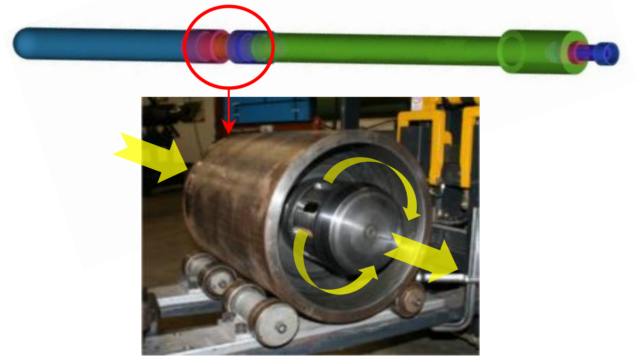

Figure 1. CAD model of the X2 reservoir, launcher and driver sections. Piston launcher station is highlighted, and the real device is shown. In the blanked off test, the end of the driver is closed. The flow has to pass from the reservoir to the driver through the launcher inside slots as shown, creating a pressure loss.

Since L1D models use a unique incompressible pressure loss factor, its determination should not depend on the mass flow therefore a stationary approach is going to be employed at a first glance.

Two type of meshing technique are employed: unstructured tetrahedral, and structured multi-block hexahedral grids, see Figure 2. The tetrahedral is later complemented with prism layers to capture boundary layers, and then converted to polyhedral for faster convergence and lower numerical diffusion. 2D models contain around $2.3\text{x}10^5$ cells. 3D hexahedral has $4.5\text{x}10^6$ cells, and unstructured polyhedral around $3.0\text{x}10^6$. No attempt was made in terms of a formal convergence analysis since the calculation is exploratory.

The 2D axisymmetric model cannot represent accurately the launcher slot geometry, one has to decide if maintaining the section shape or correct it to take into account the fact that the area grows with the radial distance. In this case the second option was taken.

A standard commercial CFD software package (ANSYS Fluent [2]) was used. The turbulence model employed is realizable k−ε which is the recommended one in case massive separated flow regions are present [2], as it is suspected. The energy equation is solved since the air is treated as perfect gas. In this case the process is assumed to be adiabatic for the walls. At the inlet total pressure is fixed to reservoir total pressure. k−ε at the inlet are obtained through fixing a turbulence intensity of 5%, and characteristic length of 1/10 of the radius, as a rough approach to fully developed turbulent flow. Static pressure is then reduced at the outlet to allow mass flow at the launcher. Pressure difference at the exact same L1D pressure loss model locations is monitored to assess convergence.

Launcher model Results

An initial case was set up for comparison of the models. An arbitrary mass flow was selected so that the mean velocity at the reservoir was about 10m/sec and the model was assumed incompressible. Table 1 shows the predicted values of head pressure losses compared to L1D losses. CFD values are calculated using 1st and 2nd order spatial accuracy [2], as a basic sensitivity analysis. Here has to be noted that the values used in L1D come from blanked off testing.

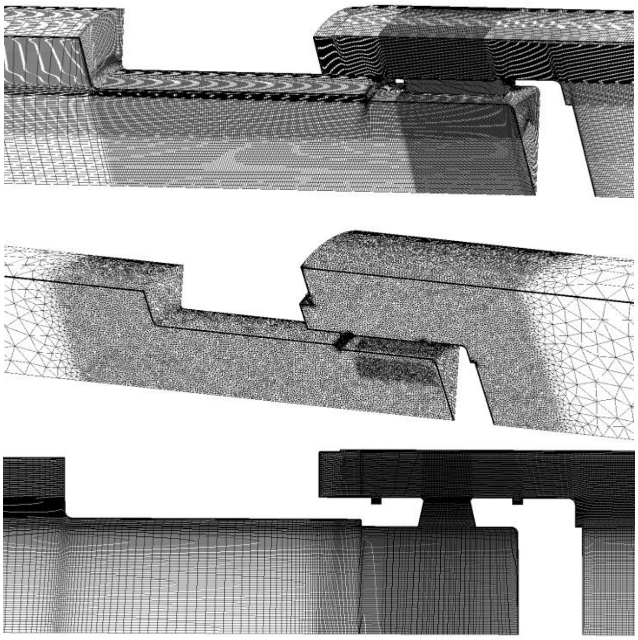

Figure 2. X2 Launcher mesh models. Structured multi-block (top). Unstructured tetrahedral and boundary layer prisms (centre), and structured multi-block 2D (bottom).

	Hexa 1st	Hexa 2nd	Tetra 1st	Tetra 2nd	2D 1st	2D 2nd
P_0 (pa)	2.2E5	2.8E5	1.9E5	2.1E5	2.3E4	2.2E4
q (pa)	6.5E4	6.8E4	6.7E4	6.7E4	9.6E3	9.7E3
K	3.43	4.08	2.82	3.17	2.47	2.31
%Error	11%	32%	-9%	2%	-20%	-25%

Table 1. Pressure loss factor $K = \frac{P_{02}-P_{01}}{\frac{1}{2}\rho v^2}, , q = \frac{1}{2}\rho v^2$, using 1 and 2 stations of the L1D model [4], (X2-LWP-2.0mm test). Reservoir Pressure loss coefficient reported there to fit experiments is 3.1. Pressure base level is 6.85Mpa.

Figure 3 shows velocity contours in a longitudinal cut of the launcher area. It can be seen that combining the area reduction and the formation of a jet at the exit of the launcher, the maximum launcher slot to inlet speed factor can easily reach 10. In the piston trajectory, peaks of mass flow higher than the ones simulated are expected, so it was suspected that the flow speed at the launcher in a real case would approach sonic conditions

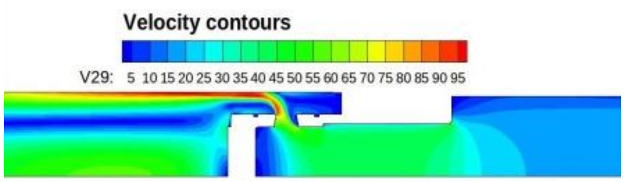

Figure 3. Longitudinal cut along slot symmetry plane showing velocity contours. Flow comes from the right.

Following this idea, the incompressible assumption was abandoned, and the most accurate model (tetrahedral, k−ε), 2nd order) was run for different mass flows. These calculations were expected to understand when the choking of the launcher occurs, results are reported in Table 2.

Two useful quantities are calculated, the critical area for choking (taking the launcher area as the minimum area, equation 1), and compressible mass flow using a discharge coefficient (area reduction), equation 2.

$$\frac{A^*}{A} = M\left[\frac{2+(\gamma-1)M^2}{\gamma+1}\right]^{-\frac{\gamma+1}{2(\gamma-1)}} \quad (1)$$

$$\dot{m} = A\,CD\,\frac{\sqrt{\gamma}P_0}{\sqrt{RT_0}}\,M\left[1+\frac{\gamma-1}{2}M^2\right]^{-\frac{\gamma+1}{2(\gamma-1)}} \quad (2)$$

P(Pa) Reservoir	P(Pa) Driver	$\dot{m}$ (Kg/s)	A*/A	Cd	K	M_{eff}
-2.6E3	-1.4E5	5.0	0.15	0.48	2.14	0.32
-1.0E4	-5.7E5	10.0	0.31	0.48	2.25	0.64
-2.3E4	-1.4E6	15.0	0.46	0.48	2.50	0.96
-3.3E4	-2.4E6	18.0	0.55	0.48	2.87	1.15
-3.7E4	-2.8E6	19.0	0.58	0.48	3.16	1.22
-4.1E4	-3.7E6	20.0	0.62	0.48	4.19	1.28
-4.2E4	-4.5E6	20.2	0.62	0.48	6.26	1.29
-4.2E4	-5.2E6	20.2	0.62	0.48	7.16	1.29

Table 2. Cases of increasing mass flow until choking of the launcher occurs. Base pressure level is 6.85Mpa.

Values found for the discharge coefficient reduce the area to around 48% of the nominal value. An effective maximum *Mach* number can be calculated using this area reduction. It can be seen than choking conditions are reached for case 3, at an inlet velocity as low as 24m/sec. Therefore, since piston velocity in a real experiment in X2 can reach over 200 m/sec it is highly suspected that the launcher is choked for a substantial part of the piston trajectory.

A possible explanation for the apparent variation of the test fitted pressure head loss coefficient can be inferred from these results. As figure 4 shows, calculated incompressible loss factor grows abruptly close to sonic conditions due to the fact that the pressure loss does not depend anymore on pressure difference but stagnation upstream conditions. If the loss model is formulated in terms of a compressible discharge coefficient, table 2 shows that its variation is under 1%.

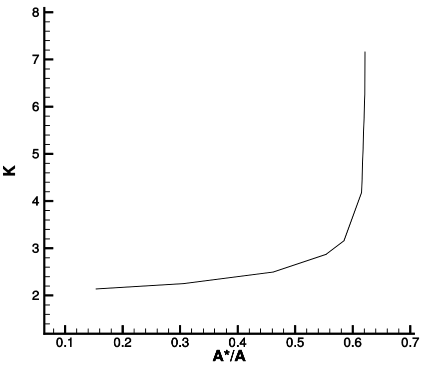

Figure 4. Pressure loss vs. critical area ratio for Table 2 cases

X2 blanked off test simulation

Once the CFD launcher model has been validated, may be possible to develop a CFD model that takes into account piston dynamics. The model used was 2D axisymmetric since the pressure loss values reported in Table 1 are not so far from the experimental ones, and the model is computationally much cheaper. All the model settings discussed in the previous section is replicated here, and test case is X2-LWP-2.0mm from [4].

The 2D mesh covers the entire reservoir, launcher and driver. It contains a moving piston simulated by the *layering* technique [2]. In essence *layering* introduces new cells in the volume behind the piston domain, and removes cells from the front domain of it. Using integrated wall forces over the piston, a 1D newton model is solved, and piston position and speed is calculated. This information is fed into the re-mesher thus closing the coupled calculation loop. Figure 5 shows a mesh picture when the piston has advanced enough to clear the launcher area.

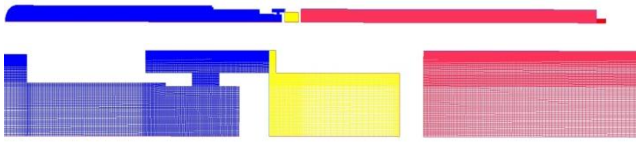

Figure 5. Mesh domains (reservoir-launcher, driver and piston, top). Details of the launcher area when piston clears the launcher exit, bottom.

The setup was solved using Fluent13 [2]. Fixed one order time-step implicit scheme ($1x10^{-6}$sec) was chosen. This approximately gives a CFL close to 1. $5x10^4$ time-steps were run ($5.0x10^{-2}$ secs) and took approximately 4 days in an 8 processor desktop PC. No attempt has been done in studying computing efficiency. The time history of Mach number contours is shown at Figure 6. Most remarkable is the strong supersonic jet coming out of the launcher from the very beginning of the trajectory. As it was precluded in the previous section, the exit of the launcher is choked for most of the piston trajectory. The formation of a shock wave at the back of the piston when moving backwards is also seen. This is not expected from a 1D model, and occurs due to the strong radial variation of the flow. Close to the axis a low speed recirculating flow exist, while at the outer radius a jet is encountered (ether supersonic of subsonic depending on the axial location).

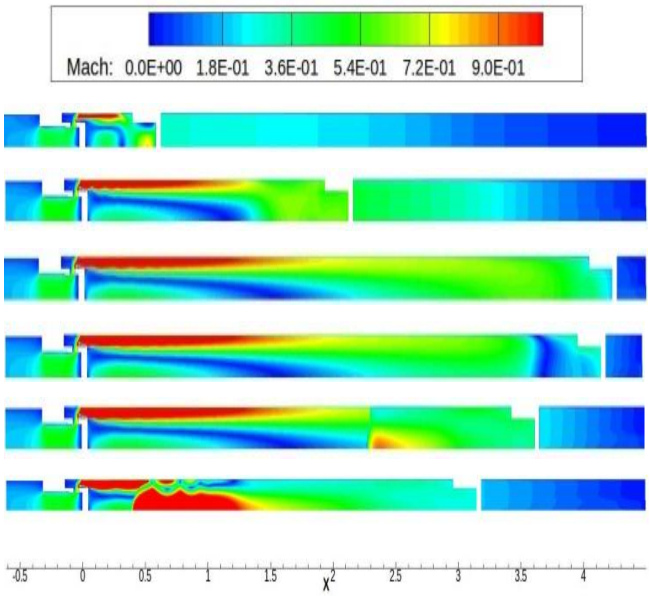

Figure 6. Time evolution of piston dynamics for arbitrary time steps, at the driver tube. Radius is scaled for a better view. Flow Mach number is selected for the contours.

Using the L1D model that fits the experimental results from [4], piston trajectory and pressure at the end of the driver tube can be compared. This is done in figure 7 and 8. It is remarkable that in figure 8, static pressure compression and expansion waves around the mean value are reasonable well captured by the 2D CFD model. These waves may introduce significant flow features in a future model where diaphragm would be allowed to burst and propagate waves downstream up to the test section.

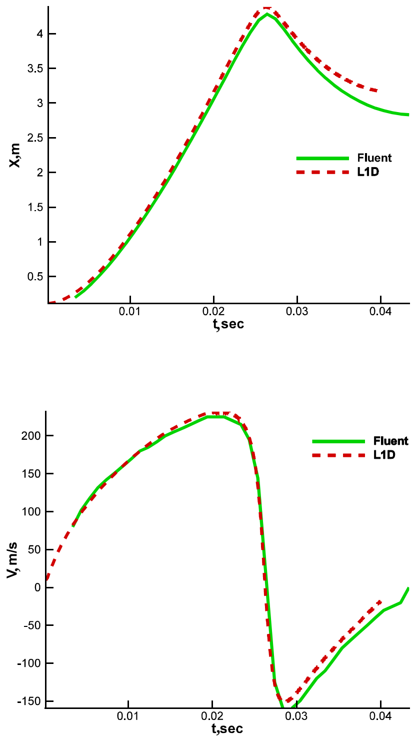

Figure 7. Piston position and speed from CFD (Fluent) model, and experimentally fitted L1D model.

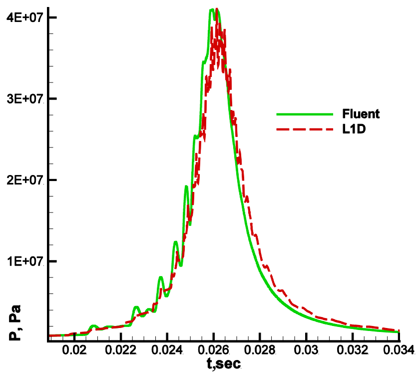

Figure 8. Static pressure trace at the end of the driver tube from CFD (Fluent) and experimentally fitted L1D model.

Conclusions

Implementation of the piston dynamics is one of the main challenges in the path towards the possibility of simulating a free piston expansion tunnel.

This paper firstly has explored the ability of CFD models (either in 2D or 3D) to correctly predict the pressure loss that occurs at the X2 launcher station. This step is previous to the implementation of any piston dynamics model, since the pressure at the back of the piston has to be accurately calculated. The main conclusion of this investigation is that CFD can predict this value with reasonable accuracy. Moreover, detailed exploration of the CFD solution can shed some light on the processes that can affect the performance of the facility. Primarily, that the launcher station may choke the flow much sooner than expected due to the formation of sonic jets, effectively reducing the area available. Since the mass-flow available to the back of the piston will be restricted by this phenomenon, the performance of the facility would get reduced. This effect may explain why different pressure loss factors are measured experimentally depending on the condition.

A 2D CFD model of the X2 driver including piston dynamics was developed. The model was validated using blank off tests reported in [4]. Results confirm the choking of the launcher, and reproduce the static pressure traces at the end of the driver. The Piston trajectory also matches accurately the experimentally fitted L1D models reported there.

Since CFD models that simulate the compressed driver gas up to the test section were reported (for example in [5]), and since implementation of piston dynamics in 2D or 3D was one of the main issues towards the accurate description of the driver gas, this proof of concept modelling exploration has shown that it is possible and feasible to perform a complete model of the whole facility by combining these two techniques. The subsequent recommendation is to develop such a model and apply its capabilities to the simulation of X2 and X3 facilities.

Acknowledgments

The authors would like to acknowledge the Australian Space Research Program and the University of Queensland for their funding in support of the Scramjet-based Access-to-Space Systems (SCRAMSPACE) project.

References

[1] Anderson, G.Y., McClinton, C.R., Weidner, J.P., Scramjet performance, in *Progress in Astronautics and Aeronautics* 189, chapter 6, 369-446, AIAA 2000.

[2] Ansys Fluent 12.0 User's guide. Ansys inc.

[3] Fry, R.S., A Century of ramjet propulsion technology evolution. Journal of Propulsion and Power, 20(1), 2004, 27-58.

[4] Gildfind, D.E., Morgan, R.G., McGilvray, M., Jacobs, P.A., Stalker, R.J. & Eichmann, T.N. Free-piston driver optimisation for simulation of high Mach number scramjet flow conditions. *Shock Waves*., **21**, 2011, 559-572.

[5] Gildfind, D.E. Development of High Total Pressure Scramjet Flow Conditions using the X2 Expansion tube. PhD Thesis, Centre for Hypersonics, Department of Mechanical Engineering, University of Queensland, Australia, 2012.

[6] Jacobs, P.A., L1d: A computer program for the simulation of transient-flow facilities. Report 1/99, Department of mechanical Engineering, University of Queensland, Australia, 1999

[7] Jacobs, P.A., Gollan, R.J., The Eilmer3 code: User guide and example book. Report 7/08, Department of mechanical Engineering, University of Queensland, Australia, 2012

[8] McGilvray, M. Scramjet testing at high enthalpies in expansion tube facilities. PhD Thesis, Centre for Hypersonics, Department of Mechanical Engineering, University of Queensland, Australia, 2008

[9] Morgan, R.G., Free-piston driven expansion tubes. *Handbook of shock waves,* editors Ben-Dor, G., Igra, O., Elperin, T. , Elsevier, 2001, **1** 603-622.

[10] Paull, A., Stalker, R.J., Test flow disturbances in an expansion tube. Journal of Fluid Mechanics. **245**, 2001, 493-5