Design of fluid-structure interaction experiment in the supersonic wind tunnel at UNSW Canberra

Zain Ahmad

The University of New South Wales at the Australian Defence Force Academy

The supersonic flow domain is one of the most researched domains of aerodynamics due to the increasing interest in high speed weapons and cruise vehicles tests. UNSW Canberra has a supersonic wind tunnel that has been widely used for static experiments. The aim of the project was to determine whether this facility can be used for aeroelastic investigations. Numerical methodology was adopted to check the feasibility of conducting a flutter experiment in the tunnel at Mach 2. Initially, a complete characterization of the tunnel was performed for Mach 2 flow using Ansys Fluent. The obtained results were validated against analytical results and previously conducted experiments in the UNSW Canberra tunnel. After that, an Aluminium panel, which was representative of a high-speed vehicle skin panel, was placed at the top of a cavity in a support model. This set-up was simulated in the supersonic wind tunnel and the response of Aluminium panel was calculated. It was observed that if the free-stream flow is considered only (disregarding the initial transient phase of tunnel), the aluminium panel with dimension of 32×90 mm undergoes flutter for panel thicknesses between 0.2 mm and 0.5 mm. If the panel thickness is increased more than 0.5 mm, flutter is not experienced by the panel, whereas, if the panel thickness is decreased less than 0.2 mm, the stress experienced by the Aluminium panel is greater than the yield stress of the material. On the other hand, when the load from the initial transient phase of the tunnel, which is characterized by extreme high pressure, was mapped on the Aluminium panel, it was observed that stress on the Al panels exceed the yield stress of the material. Hence, it was concluded that a protection mechanism is required to conduct the flutter experiment with the simulated experimental set-up.

Contents

I. Introduction 1
II. Project scope 2
III. Aim 2
IV. Summary of relevant studies 2
V. Scaling methodology for panel 4
VI. Inviscid characterization of tunnel 5
VII. Viscid analysis of model in tunnel 6
VIII. FSI simulations 9
IX. Conclusion 10
X. Future work 11

Acknowledgements 11
References 12

APPENDICES
Appendix A. Requisite Data about wind tunnel at UNSW Canberra A1
Appendix B. Non-dimensional parameter for 0.2 mm thick Al panel A2

I. Introduction

There is a worldwide interest in high-speed flight. The contemporary world is experiencing the exciting innovation of supersonic weapons and supersonic cruise vehicles. At such high speeds, fluid-structure interactions (FSI) play major role. One branch of FSI is dynamic aeroelasticity, which is the study of interaction among aerodynamics, elastic and inertial forces acting on a vehicle component [4]. Flutter is a type of dynamic aeroelastic
instability, which arises due to non-linear structural behavior of panels, as for example those used in the body of vehicles [10]. If the structural damping is not large enough, flutter can result in structural failure due to fatigue[16]. That is why flutter has always been considered a critical aspect of the flight dynamics of aircraft operating in all regimes: subsonic, supersonic and hypersonic.

UNSW Canberra has a supersonic wind tunnel, where FSI experiments have never been conducted. This facility has primarily been used for the static experiments. This has forced UNSW Canberra staff to rely on the facility of University of Southern Queensland (TUSQ) for supersonic and hypersonic FSI experiments. To enable local testing and investigate if the facility is suitable for experiments, this project aimed to determine the feasibility of the supersonic tunnel at UNSW Canberra for FSI experiments. The project performed numerical study of a simple panel flutter experiment suitable for this facility.

In this report, the scope of the project will be discussed in Section II while the aim of the project will be elaborated in Section III. Section IV will be a summarized review of literature and Section V will encompass the scaling methodology of the panel. The next three sections will focus on the results obtained during the numerical simulations. Section IX will present the conclusion. Finally, Section X will provide a brief outline for future study.

II. Project Scope

The project focused on assessing the feasibility of FSI flutter experiments to be performed in the UNSW Canberra SuperSonic Tunnel (SST) at Mach 2. Numerical study was performed to assess the feasibility of the experiment. SST characterization was conducted via numerical analysis (CFD and FEM) and required parameters (Temperature, Pressure and Mach number) were obtained in test section of SST. These parameters were used to map the load onto the testing article that was representative of the panel used in the high-speed vehicles.

III. Aim

The aim of the project was to determine whether the supersonic tunnel at UNSW Canberra has sufficient ‘steady’ flow time (after initial transient) to conduct a steady state flutter experiment that simulates behaviour of a panel (representative of high-speed vehicle skin panel) at Mach 2.

A. Challenges to achieve the aim

Three main challenges were faced to achieve the aim.

1. Time constraint
   It was decided to simulate two steady state flutter oscillations in the steady flow time of SST i.e. 20 seconds. If two oscillations were obtained in the steady flow time, we could observe if the behaviour of the panel is leading towards stability or instability.

2. Size restriction
   Test section of tunnel is 155x90 mm. If the designed panel is too small, it can become inherently too stiff to be susceptible of flutter.

3. Transient stage
   There exists a transient phase at the start of the tunnel run. This phase is characterized by high pressure, which means that the panel could undergo plasticity or fail under the adverse pressure gradient. Moreover, we wanted to make sure that if flutter was experienced by the panel, the onset of flutter occurred in the steady flow time which could be initiated in the initial transient phase otherwise.

IV. Summary of relevant studies

A. Supersonic wind tunnel at UNSW Canberra

The SST at UNSW Canberra is a medium sized wind tunnel with a cross sectional area of 155x90 mm and a field of view with 140 mm diameter. The tunnel provides useful flow time of approximately 20 seconds with an initial start-up time of approximately six seconds as shown in the experimental results of Vikram (Figure 12). Tunnel can be operated efficiently at Mach 2, Mach 2.4 and Mach 3. Dr. Sudhir Gai is the principal designer of the wind tunnel [19].

The wind tunnel facility consists of a compressor plant, a high pressure reservoir, a control valve plus control circuit, a stagnation or settling chamber, test section, second throat, subsonic diffuser and a muffler. Detailed description of wind-tunnel is shown in Appendix A. The high-pressure reservoir is filled with dried air from compressor plant to a pressure of 1400 kPa. A pneumatically actuated control valve is used to vent the compressed air to the settling or stagnation chamber. Stagnation chamber is then kept constant through control valve. The air passes through a series of wire meshes in the stagnation chamber to the test section. The wire meshes render the flow uniform. Large scale disturbances are broken up into small scale turbulence by using wire meshes and hence dissipation of turbulent energy is increased [2].

B. Blockage ratio in supersonic wind-tunnels

While conducting any experiment in supersonic wind tunnels, ‘blockage ratio’ is considered to be a very important factor [5]. Because of blockage, the flow experiences a smaller effective area and choking conditions
(Mach = 1) can be reached in regions within the core flow. Blockage generally results in flow distortion, variation of velocity and off-designed 3-D effects [5]. Blockage ratio is defined as:

\[
\text{Blockage Ratio} = \frac{\text{Projected area of test body}}{\text{Cross-sectional area of tunnel}}
\]  

(1)

Generally, a blockage ratio under 10% is considered acceptable while blockage ratios under 5% are considered to be very safe in this regard [5].

C. Flutter in supersonic flow

The outer skin on almost all supersonic vehicles is supported by variously spaced bars or ribs which divide it into individual panels forming an array. These panels are exposed on one side to a parallel supersonic airstream and are susceptible to induced FSI called panel flutter. The prevention of panel flutter becomes a primary design criterion in some cases [15]. Figure 1 shows various panels on the tail of X-15. In this project, panel flutter of a high-speed vehicle was simulated in a free stream flow of Mach 2.

One of the primary characteristics of the flutter is the fact that it is limited to a range of speeds. Below and above the extremes of this speed range, there is aerodynamic stability [30]. This is because as we increase the speed range, the aeroelastic forces overcome the damping of the material, making it flutter. As we increase the speed further, the temperature of the panel increases causing in-plane stress and thus buckling is introduced. After buckling, the additional temperature induces corrugation in the panel, making it stiff thus stopping the flutter of the panel [14]. Another important aspect of flutter is how much time is needed to reach instability, namely the time-to-onset flutter (\(\tau\)) [7]. We know that by increasing the stiffness of any panel (e.g. increasing Young’s Modulus, thickness or changing the boundary conditions), we increase the modal frequencies and the panel is less prone to flutter. It is also known from Freydin et al. [22], however, that \(\tau\) decreases with increase in the second natural frequency i.e. \(\tau = \frac{1}{f_2}\). Thus, if the panel is not stiff enough, flutter will take place after a long time, which can be potentially longer than the test time [22]. Conversely, too stiff panels (i.e. high natural frequencies) are less susceptible to structural instability. As a consequence, panel size in our experiment has to be stiff enough to survive initial transient flow and reduce \(\tau\). Whereas at the same time, it should not be too stiff to inhibit flutter at all.

D. Different methodologies adopted for flutter analysis

Different methodologies have been adopted in literature to analyze the flutter problem. Analytical approach including Piston theory, have been very common in analyzing the flutter problem in supersonic flow [32]. From the Piston theory, simple approximation of pressure was derived as [30]:

\[
P(x,t) = \frac{\rho u}{\sqrt{\mathbf{g}^2 - 1}} \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} \right)
\]  

(2)

Where \(\rho, u, w\) and \(M\) are flow density, speed, deflection of plate and Mach number respectively. This above equation provides very accurate results for high Mach number problems but in later studies it was shown that classical Piston theory is not valid for low supersonic problems. Moreover, Piston theory cannot solve the viscous or 3-D effects of the flow[30]. Similarly, it was affirmed that local Piston theory using the mean flow methodology has been able to solve the 3-D effects but still it is not validated for low supersonic Mach numbers. Other theories like Donnel’s shell theory and Potential theory by Leonard and Hedgpet have been generated by keeping a specific problem in consideration [25]. They have generated high fidelity solutions for those particular problems but have not been affirmed to provide a feasible solution for other cases [25]. Since we were operating in low supersonic regime and viscous effects will play a role because of the cavity present in our simulated experimental set-up, taking an analytical approach would not have been able to provide accurate results for our case. That is why we decided to use numerical methodology, which will be more accurate for our case and will be cost efficient than actual conduct of experiment. Autodesk Inventor was used for modelling, ICEM and Ansys were used for meshing, Ansys Fluent was employed for CFD characterization, static and transient structural components of Ansys were used to test the panel in FSI simulations.
V. Scaling methodology for the panel

Figure 2: Schematic of two-sided clamped panel at the top of cavity of the support model

Figure 2 shows the boundary conditions of the support model and testing panel while Figure 3 represents the schematic of the experiment in the wind tunnel. We can see in Figure 2 that support model is held by fixed supports from four sides, whereas the panel is clamped on two sides and is placed at the top of cavity of the model. This geometry was chosen after careful consideration of the fact that panels used on the body of high-speed vehicle are usually clamped and they also have a cavity underneath them [7]. Before proceeding with our simulations, we decided the non-dimensional constants that dictate the scaling between actual vehicle panels and testing article in the supersonic tunnel.

A. Kordes Parameters

From literature we extracted that Kordes et al. [14] studied the panel flutter of an X-15. They performed the experiments for panel flutter in the same operating range and boundary conditions as ours i.e. clamped on two sides and Mach 2 flow horizontal to the panel. Their analysis is regarded as one of the canonical cases for the panel flutter of supersonic flow [14]. Its theoretical results have been compared with the practical results for the X-15 panels in the past, showing good agreement [14]. They chose the flutter parameter (a non-dimensional constant), which we named as Kordes parameter, as \((\sqrt{M^2 - 1} \times \frac{E}{q} \times \frac{t}{l})\) displayed on vertical axis of the Figure 4. In this parameter, \(M\) and \(q\) represent the Mach number and dynamic pressure of flow in test section, whereas, \(E, t\) and \(l\) are Youngs Modulus, thickness and length of the material respectively. When analyzed against other similar studies [29], we found out that most of the parameters were taken into consideration while assigning Kordes parameter but position of center of gravity was not considered. Since the testing panel is clamped on two sides and flow is horizontal to panel, center of gravity is not a dictating factor for our case [29]. Thus, Kordes parameter is considered as first non-dimensional constant for scaling of the panel.

The bars in Figure 4 represent the scatter of results (Kordes parameter) for changing different parameters \((M, q, t, l, E)\) when aspect ratio \((\frac{\text{length}(l)}{\text{width}(w)})\) on x-axis was kept the same. From Kordes parameter we see that increasing Mach number(M), dynamic pressure(q) and panel-length(l) promotes flutter, whereas increasing the...
Elastic Modulus \((E)\) or panel-thickness \((t)\) inhibits flutter. The relationship between Kordes parameter and aspect ratio of the tested panel in Figure 4 provided insight for the selection of geometry in our case.

B. Ratio of modal frequencies

Modes are analogous to the degrees of freedom (DOF) of a body, e.g. for a rigid body there are 6 DOFs. In panels and plates there are an infinite number of natural modes. Generally, the first 6 modes/modal frequencies are enough to capture the dynamics of the panel [4]. If we apply boundary conditions to a rigid body, the latter loses one or more DOFs. A panel, instead, shifts its own natural modes towards higher frequencies. Higher frequencies normally translate into high internal stress or small displacement. External energy is not converted into work, but it is stored as elastic energy [4].

One of the mechanisms of flutter is coalescence of the first two modes, which are bending and torsional modes normally. Particularly, flutter depends on the ratio of first and second modal frequencies \(\left( FR = \frac{f_2}{f_1} \right) \) [4]. Values of FR close to 1 promote flutter, thus it is important to correctly evaluate the natural frequencies of the panel [4]. The ratio of the first two modal frequencies was the second non-dimensional constant chosen for scaling between panel of supersonic vehicles and our testing article.

C. Ratio of deflection over length

Another important flutter characteristic is deflection magnitude. Because of space restrictions, the panel size will be limited and thus the deflections will be generally small. We have to measure the displacement history which can be evaluated with the current measurement techniques, namely laser scanning (accuracy of few tenths of micron) and schlieren photography (accuracy of the order of millimeter). However, it is to be noted that it is a non-dimensional constant i.e. deflection over length ratio that is of interest for our case.

The testing panel in the tunnel will be representative of the actual vehicle panel if both have the same Kordes parameter and ratio of modal frequency. Similarly, deflection over length ratio of the testing article will be representative of the flutter for the actual flight vehicle panel. Once, scaling parameters were decided, inviscid characterization of the tunnel was performed which will be discussed in next section.

VI. Inviscid Characterization of tunnel

A three-phase approach was adopted to perform the inviscid characterization of the supersonic tunnel i.e. modelling, meshing and analysis. It is to be noted that inviscid characterization was necessary in order to understand the operation of the tunnel in its easiest form. This provided the baseline for the further analyses to be performed.

Initially, two symmetry planes were used to divide the tunnel into four parts and one of the parts was used as the domain in order to save the computational cost. ICEM-CFD was employed to mesh the domain. Orthogonal grid lines were used in the sub-sonic region and diffuser region of the tunnel, whereas, curved grid lines were used to capture the sonic line of the flow as shown in Figure 5. The grid density was increased in the test-section area to capture the flow.

![Mesh of quarter tunnel in ICEM-CFD](image)

*Figure 5: Mesh for the inviscid characterization of quarter of tunnel*
After meshing the domain, Fluent was used to analyze fluid dynamics. Inviscid-transient analyses were performed, and the resulting Mach number and Pressure were captured. Figure 6 and Figure 7 show how free-stream pressure and free-stream Mach number vary along the length of the tunnel respectively. We can observe that Mach number increases along the length of the tunnel and we obtain Mach 2 in the test-section. Similarly, the pressure in the stagnation chamber is held constant at 220 kPa which decreases along the length of the tunnel and we obtain a constant 27.6 kPa of pressure in the test section.

![Figure 6: Pressure distribution for free-stream inviscid flow along the tunnel length](image1)

![Figure 7: Mach distribution for free-stream inviscid flow along the tunnel length](image2)

**A. Verification and validation of results for inviscid characterization**

Before proceeding to viscid characterization, the CFD results of inviscid case were validated against the analytical results of Mach number and pressure distribution in the test section. Figure 9 shows how the isentropic equations result in Mach 2 flow and 27.3 kPa of pressure in the test section. This is approximately what we obtained from our CFD calculations as shown in Figure 6 and Figure 7. We also performed the mesh independence study for the Pressure at a point in test section (1, 0, 0.045) m, and it was concluded that 17010 grids with mesh modification as shown in Figure 5 give us the acceptable result. From the pressure plots, it was observed that if we further increase the mesh density, the error in pressure data is less than 2% (Figure 8), which clearly signifies that the study is independent of grid size.

Once inviscid characterization of the tunnel was completed, we focused on the viscid-transient characterization of tunnel and the actual simulation of the experiment, which will be discussed in next section.

**VII. Viscid Analysis of Model in tunnel**

Viscid analysis was necessary because it simulated the actual operation of the tunnel where viscosity plays a role due to the presence of cavity. Viscous analysis of the experiment in tunnel was also simulated in Fluent.
Fluent is a computational fluid dynamics software package used to model flow, turbulence, heat transfer and reactions. It uses equations of conservation of mass and momentum for all flows. Since we were dealing with compressible flow, we had to employ conservation of energy equation. As our flow was in the turbulent regime, additional transport equations were also used. We used a density based transient simulation with SST k-ω model. Standard SST and k-ε are multiplied with a blending function and both models are added together in order to generate SST k-ω. This blending function activates k-ω model near wall regions and activates k-ε model away from the surface. SST k-ω model incorporates a gradual change from standard k-ω model (in inner region of boundary layer) to higher Re Number version of k-ε model (in outer part of boundary layer). As a result, SST k-ω model behaves more accurately both near wall and far field zones and can be applied for wider class of flows, including supersonic flow. Time step size used in Fluent simulations was $1 \times 10^{-3}$. Residual of at least $1 \times 10^{-3}$ was ensured for every parameter. Second order upwind was employed in all conducted simulations to bring more accuracy.

A. Viscid Transient Analysis of Tunnel

For the first part, viscid-transient simulation of supersonic tunnel was performed to see the difference between viscid and inviscid case. Figure 10 shows the fully established viscid flow. The representation of different regions of the flow in the tunnel is clearly evident in Figure 8. We are able to identify the boundary layer region in the test section. One important thing to note in viscid simulation is its difference from inviscid case. We can see in Figure 10 that the test section Mach number is approximately 1.85 unlike Figure 7 where test section has Mach number of 2. This is due to the viscous aspect of the flow. Since we have walls around the flow, the viscous losses occur which result in the decrease of Mach number. It can be seen in Figure 11 test section experiences 28 kPa instead of 27.3 kPa as observed in the inviscid case (Figure 6).

B. Validation of results for viscid characterization

As the results from viscous-transient simulation were going to be used further in the project, it was necessary to confirm that the results obtained represent the actual operating wind tunnel. We validated our pressure-transient

![Figure 10: Mach number distribution in tunnel](image1)

![Figure 11: Pressure distribution in tunnel](image2)

![Figure 12: Validation of Pressure-transient obtained from simulation against experimental results](image3)
against one of the experiments conducted in the UNSW Canberra wind tunnel by Vikram Sridhar [21]. We can see in Figure 12 that our simulated free-stream pressure in the test section is 28 kPa and the free-stream pressure obtained for the same test section area via experimentation is 27.7 kPa. This validated our results for the free-stream pressure. For the initial transient phase, we can observe that it lasts approximately 7.1 s for simulated case and approximately 6 s for the actual experiment. This, however, does not affect the validity of results for free-stream condition. One important thing to note here is that we simulated the experiment with 200 kPa of stagnation pressure, whereas experiment was conducted with a stagnation pressure of approximately 100 kPa. This also does not affect the validity of our results as both operating conditions are used in SST to perform experiment and simulate Mach 2 flow.

We performed a mesh independence study for viscid simulation. A more detailed mesh was required to capture the boundary layer and hence number of required grids for acceptable solution increased. To obtain error less than 2% in pressure data (as shown in Figure 13) at a point in test section (1, 0, 0.045) m, 124094 grids were required. It is to be noted that full tunnel is modelled in viscid simulation unlike inviscid case where quarter of the tunnel was modelled.

B. Simulation of experimental set-up

Support model with panel (at the top of cavity) being placed into the tunnel was simulated next, and flow distribution was analyzed in viscid flow. Refinement in meshing was required to obtain the required results. The refined mesh for capturing of boundary layer and the flow inside the cavity can be seen in Figure 16.
In Figure 15, it is shown that an Aluminum panel with dimensions of 32×90 mm and clamped at two sides is placed at the top of the cavity of a support model. The distribution of pressure and Mach number at a mid-plane (i.e. between two walls) can be seen in Figure 14 and Figure 15 respectively. We can clearly observe in Mach number close-up that since the two sides of the cavity are open, flow is able to come into the cavity through these openings but the Mach number inside cavity is very low. If we look at pressure close-up, we observe that pressure in the cavity is higher. This produces a pressure differential between the top and bottom of the plate, helping in establishing the flutter we desire. Moreover, it is also the true representation of actual flight condition as there will be a pressure differential between the top and bottom of the panel. Once the loads from the fluid dynamics were obtained for the experimental set up, FSI simulations were performed which are explained in next section.

VIII. FSI simulations

We performed one-way FSI simulations in which Fluent transfers the data to the structural component and deforms the body meshes thus generating the resulting deflection.

The data from fluid domain was mapped onto the interested structural part i.e. panel at the top of cavity. The load mapping results for 0.2 mm thick Aluminum panel with dimensions of 32×90 mm are shown in Figure 17 and Figure 18. Figure 17 shows that the maximum stress (265 MPa) is observed at the edges of the plate, which is less than the yield strength of material (Aluminum) i.e. 276 MPa. We can see in Figure 14 that maximum deflection is observed at the middle-edges of the two-sided clamped plate. This suggests single mode flutter in the two-sided clamped panel.

We performed a number of one-way FSI simulations on different materials with different thicknesses. Aluminum was found to be the most suitable material for conduct of FSI experiment due to less stiffness when compared to other materials like steel. Figure 19 shows the results of deflection and stress on Al panels with different thicknesses. It was observed that if use any panel of thickness less than 0.2 mm, the stress being applied at the panel exceeds the yield stress of the material as shown in Figure 15, whereas, if we use a panel which is thicker than 0.5 mm the deflection obtained does not give us any flutter. Panels with thickness of 0.2 mm seemed to be the most suitable for flutter. It is to be noted, however, that the results in Figure 19 only represent the deflection and stress due to the free stream conditions of the tunnel. It does not account the initial transient phase of the tunnel.

Final Project Report 2019, UNSW Canberra at ADFA
When the initial-phase transient load was mapped on the Al panels, it was observed that the load on most of Al panels exceeded the yield strength of the material. It must be noted here, however, that load of different time steps of initial phase was mapped using the steady one-way FSI simulations, which means that fidelity of results was not very high for initial transient phase. A detailed two-way FSI is recommended to study the effects of initial transient phase with high accuracy, which was started but has not been completed in this project unfortunately. Based on low fidelity results of initial transient phase, where stress on Al panel exceeds the yield stress of the material, we can say that the flutter experiment given the current set-up can take place if the initial transient phase is mitigated via protection mechanism.

IX. Conclusion

A. Blockage ratio
As explained earlier, blockage ratio is very important consideration for an experiment that is to be conducted in supersonic wind tunnel. Equation 1 calculated the blockage ratio for our experiment which turned out to be 5.8%. This is slightly above the best possible expected blockage ratio but still it lies in the acceptable region [5]. Thus, we can say that this experiment can be performed in the tunnel without compromising blockage ratio, thus ensuring the flow is not choked.

B. Non-dimensional scaling parameters
The selection of non-dimensional parameters was done by doing a detailed literature review as summarised in Section V. These non-dimensional parameters can be calculated for any testing article and can be used for the scaling purposes. A calculation and illustration of these non-dimensional parameters for an aluminium panel 90×32 mm with 0.2 mm of thickness is shown in Appendix B. The same analysis can be performed to calculate the non-dimensional parameters of testing article with any other thickness.

C. Verification and validation of results
A mesh independence study was performed for the inviscid and viscous characterization of the tunnel and it was found out that with some mesh modification in the test section area and area of cavity, as depicted in the figures, we can verify our results. 100% of node mapping was achieved for 1-way FSI simulations of free-stream flow verifying the fidelity of the results produced. Similarly, inviscid and viscous characterization of the tunnel were validated against analytical and experimental results respectively.

D. Summarization of Results
When only free stream conditions were considered, it was concluded that Al panels with dimensions of 32×90 mm exist in the flutter region for thickness between 0.2 mm and 0.5 mm. If we exceed thicknesses beyond 0.5 mm, we enter into a non-flutter region. Similarly, if we reduce thickness less than 0.2 mm, the stress on Al panel exceeds the yield strength of the material. Thus, going back to our aim, the initial conclusion (when we only considered the free-stream conditions) is that the flutter region exists when we keep the thickness of Al panel (32×90 mm) between 0.2 mm and 0.5 mm.
However, when we mapped the initial transient loads on the Al panel, which is characterized by extreme high pressure, it was observed that the load on the panels exceeded the yield strength of the material. As explained earlier, the aspect of low fidelity of transient-phase simulations must be kept in consideration and two-way FSI must be performed (which was started in this project but could not be finished due to time restriction) in future.

Based on above discussion it is concluded that an FSI experiment, which ensures the onset of flutter in free-stream flow, using simulated experimental set-up can be performed in UNSW supersonic tunnel with some protection mechanism at the top of panel, which will be used to mitigate the initial transient phase.

X. Future work

There are multiple options to take this study further and try to broaden the ways in which the supersonic tunnel at UNSW Canberra can be used. We will briefly state a few recommendations here.

1) In this project, the main restricting factor for the conduct of FSI experiment was the initial transient phase due to its high pressure. One can employ the protection mechanism at the top of the experimental set-up for the initial transient phase and slide it once the transient phase is over. By doing so, it will be ensured that the Al panel is not affected by the initial phase and the onset of flutter is in the free-stream conditions. There are, however, a number of limitations associated to the things that can be performed with supersonic wind tunnel which means that employment of a protection mechanism might require modification in the test section of the tunnel. A possible selection of protection mechanism is shown in the Figure 20 and Figure 21, where 5 mm diameter rod and a protection plate constitute the protection assembly. This assembly will cover the panel and the cavity in the transient phase, and it will be slid upwards after the transient phase is over, such that the top of protection plate is parallel to the wind tunnel top wall minimizing the blockage ratio. This protection mechanism will also not impinge any shock on the testing article making it flutter in clean free-stream flow.

2) The SST at UNSW Canberra is a small to medium size wind tunnel. In this project, we simulated the panel flutter experiment in this tunnel. Another strategy could be to perform a simple wing flutter experiment first, which can employ a fiberglass cantilevered plate. It is known that these experiments are easy to set-up and the results obtained will be able to suggest a better avenue to mitigate the problem of transient phase.

Acknowledgements

I would like to acknowledge my supervisors Dr. Gaetano Currao and Prof. Andrew Neely for their support throughout the course of this project. I would also like to thank all members of thermofluids panel for their continuous guidance.
References

[29] T. Theodorsen, I.G. Mechanism of flutter; a theoretical and practical application of flutter mechanism, NASA.

Final Project Report 2019, UNSW Canberra at ADFA
Appendix A

Requisite Data about wind tunnel at UNSW Canberra

Schematic of supersonic wind tunnel at UNSW

Final Project Report 2019, UNSW Canberra at ADFA
<table>
<thead>
<tr>
<th>$x$-coord</th>
<th>$y$-coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>199.276</td>
</tr>
<tr>
<td>100</td>
<td>199.276</td>
</tr>
<tr>
<td>645.3</td>
<td>61.68732</td>
</tr>
<tr>
<td>670.3</td>
<td>55.86156</td>
</tr>
<tr>
<td>695.3</td>
<td>51.07655</td>
</tr>
<tr>
<td>720.3</td>
<td>47.6318</td>
</tr>
<tr>
<td>745.3</td>
<td>45.70828</td>
</tr>
<tr>
<td>770.3</td>
<td>45.35387</td>
</tr>
<tr>
<td>795.3</td>
<td>46.48288</td>
</tr>
<tr>
<td>820.3</td>
<td>48.8894</td>
</tr>
<tr>
<td>845.3</td>
<td>52.27307</td>
</tr>
<tr>
<td>870.3</td>
<td>56.27409</td>
</tr>
<tr>
<td>895.3</td>
<td>60.51358</td>
</tr>
<tr>
<td>920.3</td>
<td>64.63456</td>
</tr>
<tr>
<td>945.3</td>
<td>68.33908</td>
</tr>
<tr>
<td>970.3</td>
<td>71.41729</td>
</tr>
<tr>
<td>995.3</td>
<td>73.76526</td>
</tr>
<tr>
<td>1020.3</td>
<td>75.38974</td>
</tr>
<tr>
<td>1045.3</td>
<td>76.39946</td>
</tr>
<tr>
<td>1070.3</td>
<td>76.98429</td>
</tr>
<tr>
<td>1095.3</td>
<td>77.38475</td>
</tr>
<tr>
<td>1120.3</td>
<td>77.8557</td>
</tr>
<tr>
<td>1437.3</td>
<td>78.17663</td>
</tr>
<tr>
<td>1847.3</td>
<td>75.0</td>
</tr>
<tr>
<td>2257.3</td>
<td>80.0</td>
</tr>
<tr>
<td>3787.3</td>
<td>160.0</td>
</tr>
</tbody>
</table>

Coordinates of UNSW supersonic tunnel operating at Mach 2
Non-dimensional parameters for 0.2 mm thick Aluminium panel of dimension 32×90 mm

1) Kordes Parameter (Star shows the Kordes parameter of 0.23 for 0.2 mm thick Al panel with l/w ratio of 2.8)

2) Modal Analysis: Modal Analysis was performed for 0.2 mm thick Al panel and frequency ratio was obtained as: \( \frac{F_1}{F_2} = \frac{1093.1}{1119.7} = 0.9762 \)

3) Deflection over length ratio: Since the maximum deflection for 0.2 mm thick Al panel was 1.8 mm, the deflection over length was 0.056