Simulating Hypersonic Fluid Structural Interaction of an Inclined Cantilever Panel Using the Conservation Element Solution Element Method

[1]Luke Pollock	UNSW Canberra	L.Pollock@adfa.edu.au
[2]Graham Wild	UNSW Canberra	G.Wild@adfa.edu.au

Biography

- Research Associate – UNSW Canberra at ADFA | 2022 - Current
  - *Aerothermal vehicle shape distortion*

- Doctor of Philosophy Student (PhD Student) – UNSW Canberra at ADFA | 2021 - Current
  - *Non-destructive testing and structural health monitoring of hypersonic vehicles*

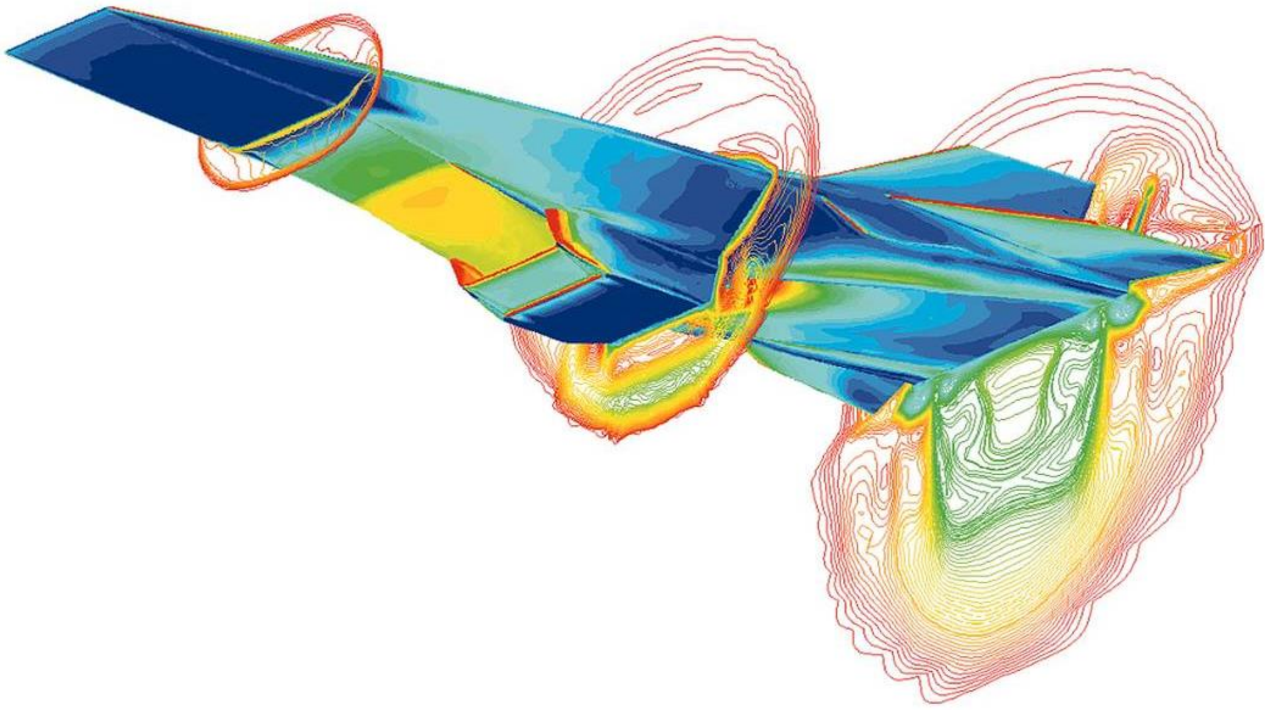
- Bachelor of Aerospace Engineering (Honours) – RMIT University | 2016 - 2020
  - *Acoustic phased arrays for structural passive health monitoring*

Hypersonic FTSI

- Low weight constraints results in thin panels prone to aeroelasticity – very important for geometry sensitive structures
  - Control surfaces
  - Inlets
- Thermal load reduces the stiffness of the material, worsening aeroelasticity
- Harmonic coupling with flow can be devastating

Hypersonic CFD

- Wide range of flow conditions
- Extremely coupled multiphysics
  - Structural interaction (FTSI)
  - Chemically reacting flows
  - Magnetohydrodynamics
- Requires massive computing and novel schemes for accuracy and efficiency



CFD of the X-43A at Mach 7

Motivation

- Validate CESE method for hypersonic FTSI

- Evaluate the potential for the development of an in-house or open source CESE code

Conservation Element Solution Element Method

- Unified treatment of space-time
  - Space and time are treated on the same footing such that a 2D domain is treated as a 3D Euclidean space.
- Modelled on the integral form of the conservation laws instead of the differential form
  - Solutions do not have to be smooth – such as the case for shocks and contact discontinuities
- Derivatives are treated as unknowns

Governing Equations

Begin with the 2D compressible Navier-Stokes equations

$$\frac{\partial \boldsymbol{U}}{\partial t} + \frac{\partial \boldsymbol{F_{inv}}}{\partial x} + \frac{\partial \boldsymbol{G_{inv}}}{\partial y} - \frac{\partial \boldsymbol{F_{visc}}}{\partial x} - \frac{\partial \boldsymbol{G_{visc}}}{\partial y} = 0 \qquad (1)$$

$\boldsymbol{U}$	$\boldsymbol{F_{inv}}$	$\boldsymbol{G_{inv}}$	$\boldsymbol{F_{visc}}$	$\boldsymbol{G_{visc}}$
ρ	ρu	ρv	0	0
ρu	$\rho u^2 + P$	ρuv	τ_{xx}	τ_{xy}
ρv	ρuv	$\rho v^2 + P$	τ_{xy}	τ_{yy}
ρe	$(\rho e + P)u$	$(\rho e + P)v$	$\tau_{xx}u + \tau_{xy}v - q_x$	$\tau_{xy}u + \tau_{yy}v - q_y$

Governing Equations

Define a three-dimensional Euclidean space E^3 with co-ordinates, x, y, t

Therefore, Eq. (1) can be written in divergence free form:

$$\nabla \cdot \boldsymbol{h} = 0, \qquad \boldsymbol{h} = (\boldsymbol{U}, \boldsymbol{F}_{\boldsymbol{inv}}, \boldsymbol{G}_{\boldsymbol{inv}}, \boldsymbol{F}_{\boldsymbol{visc}}, \boldsymbol{G}_{\boldsymbol{visc}})$$

$$\oint_{S(V)} \boldsymbol{h} \cdot \boldsymbol{n}\, dS$$

UNSW CANBERRA

Governing Equations

$$\nabla \cdot \boldsymbol{h} = 0, \;\; \boldsymbol{h} = (\boldsymbol{U}, \boldsymbol{F}_{\boldsymbol{inv}}, \boldsymbol{G}_{\boldsymbol{inv}}, \boldsymbol{F}_{\boldsymbol{visc}}, \boldsymbol{G}_{\boldsymbol{visc}})$$

$$\oint_{S(V)} \boldsymbol{h} \cdot \boldsymbol{n} \, dS = 0$$

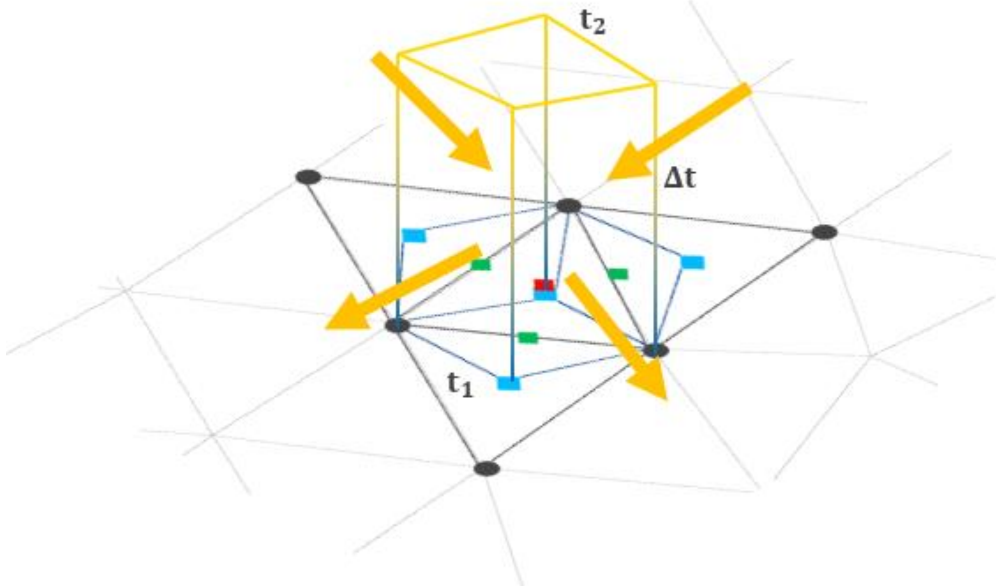
Contrast with conventional FVM where time is treated as a special dimensions

$$\frac{\partial \boldsymbol{h}}{\partial t} + \nabla \cdot \boldsymbol{h} = 0$$

$$\frac{\partial \boldsymbol{h}}{\partial t} dS + \oint_{S(V)} \boldsymbol{h} \cdot \boldsymbol{n} \, dS = 0$$

Discretisation

- A mesh is discretised into **Solution Elements (SEs)** and **Conservation Elements (CEs)**



- Values are stored at the nodes of SEs whilst flux is enforced across CEs

- The most simple 2D mesh is composed of triangles and is hence, well suited for unstructured solvers

Discretisation

- Inside each SE, a Taylor series expansion is used to approximate the flow variables
- Order of the scheme is equal to $M+1$, where M is the order of the Taylor series
- Possible to construct nth order methods

$$u_m^* = u_m + u_{m_x}(x - x_j) + u_{m_y}(y - y_j) + u_{m_t}(t - t_j)$$

Dissipative Scheme

- The core scheme is non-dissipative
- Dissipation is added by weighting the derivatives – this creates the $a - \alpha - \beta - \epsilon$ scheme

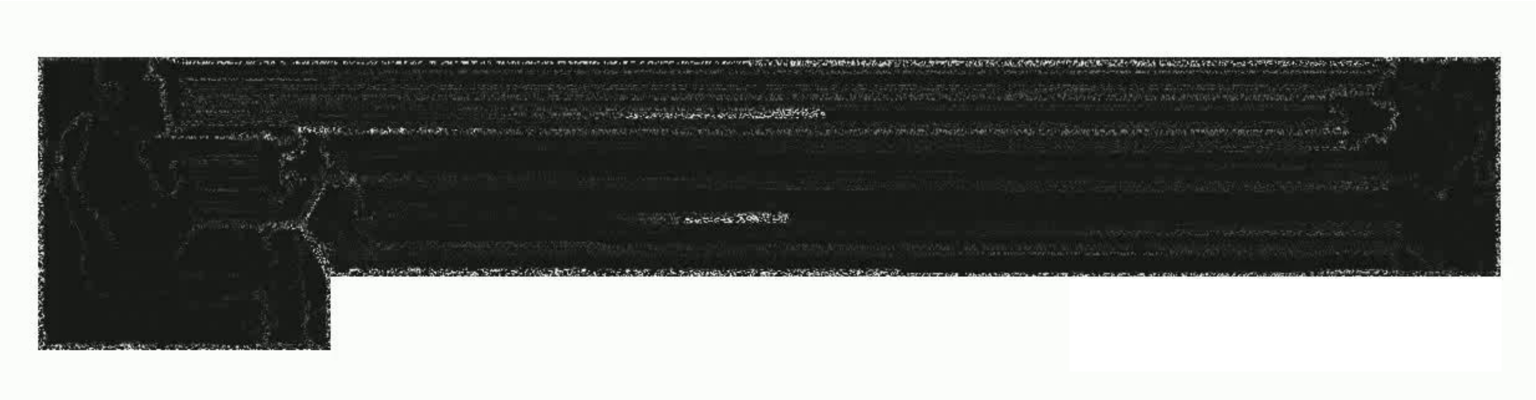
$$u_m = u_m^a + 2\epsilon(u_m^c - u_m^a) + \beta(u_m^w - u_m^c)$$

$$u_m^w = \frac{|x_+|^\alpha x_- + |x_-|^\alpha x_+}{|x_+|^\alpha + |x_-|^\alpha}$$

CESE for Hypersonics

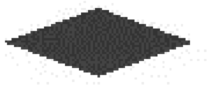
- Riemann solver-free shock capturing
- Global and local flux conservation is conserved
- High fidelity acoustic interaction (great shock interaction)
- Easily parallelized

- Has been used for:
  - MHD
  - Reacting flow
  - Multiphase flows
  - Detonation
  - Aeroacoustics



CESE for Hypersonics

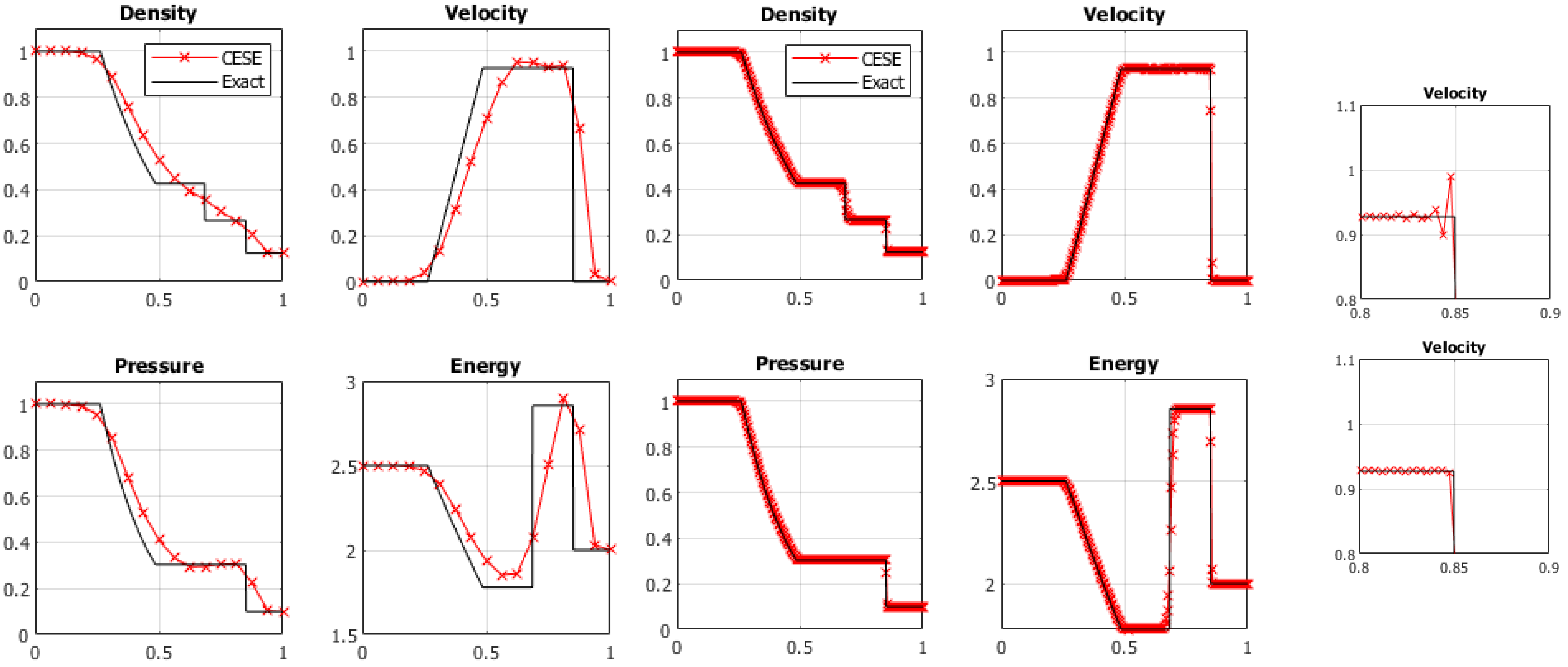
- Accurately and crisply resolves discontinuities
- Stability only depends on CFL
- Approximately 33% more efficient than conventional FVM
- Easy treatment of boundary conditions (great non-reflecting BCs)
- Excellent at handling stiff terms (no Reimann solver makes this easier)



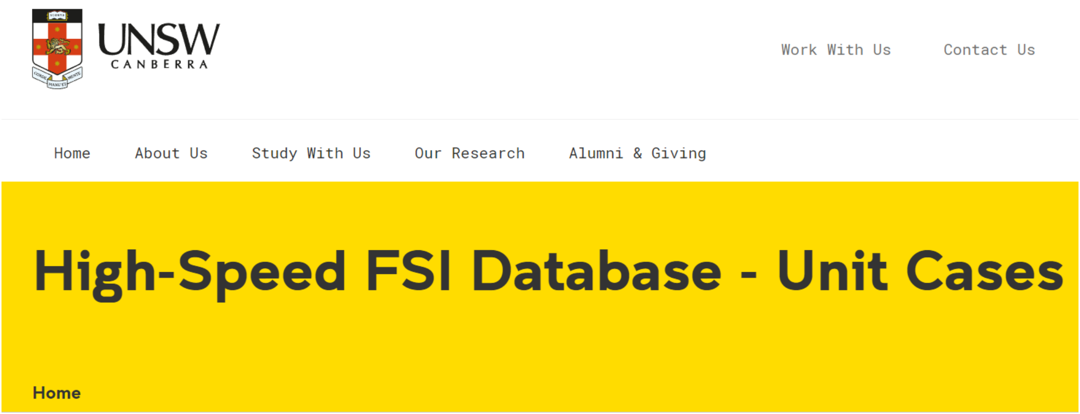
Ongoing Work

- CFL invariant schemes
- Turbulence modelling
- Solution acceleration (multigrid)
- Implicit time stepping
- Improved dissipative models
- Higher order schemes

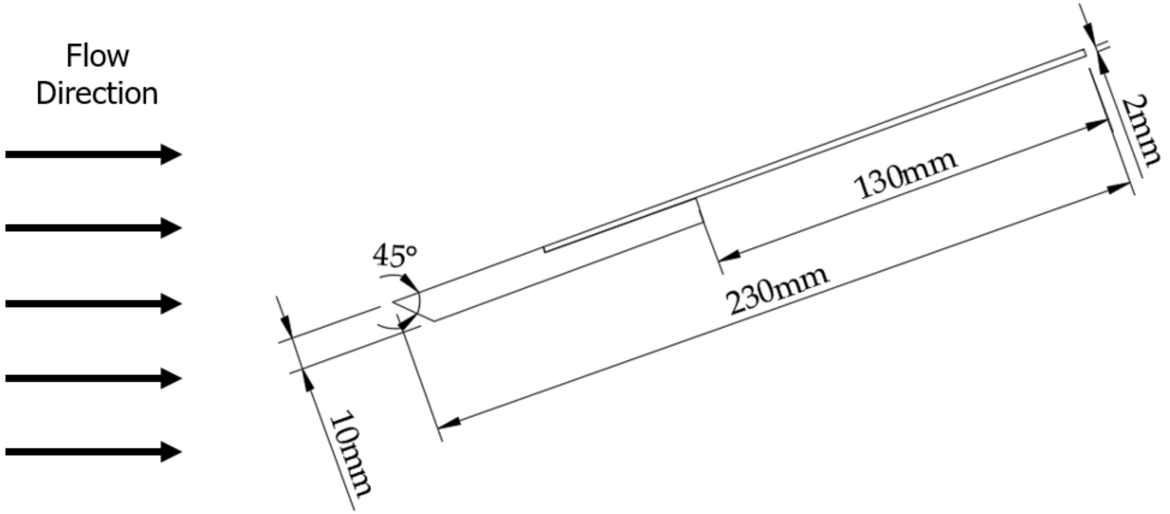
Sod’s Shock Tube



FSI Validation Test Case



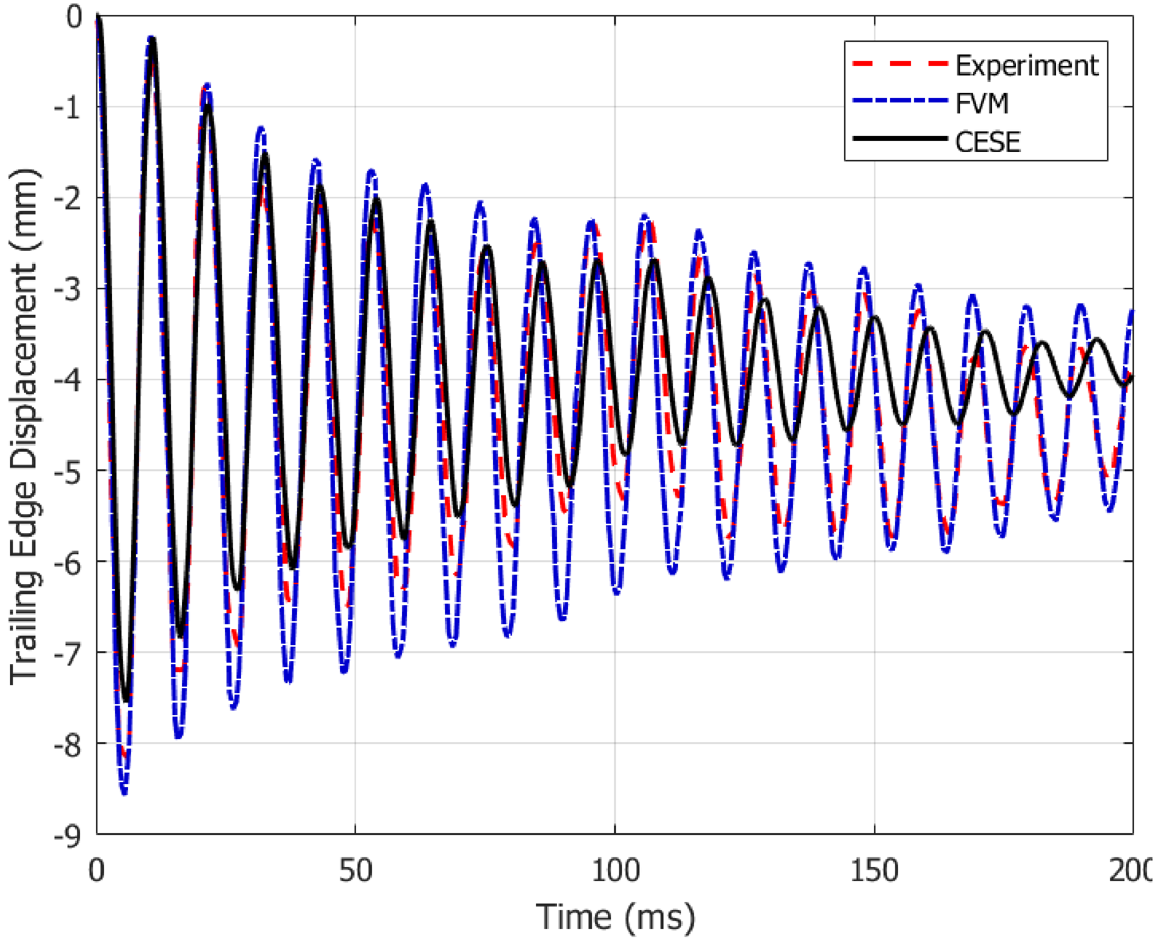
https://www.unsw.adfa.edu.au/high-speed-fsi-database-unit-cases

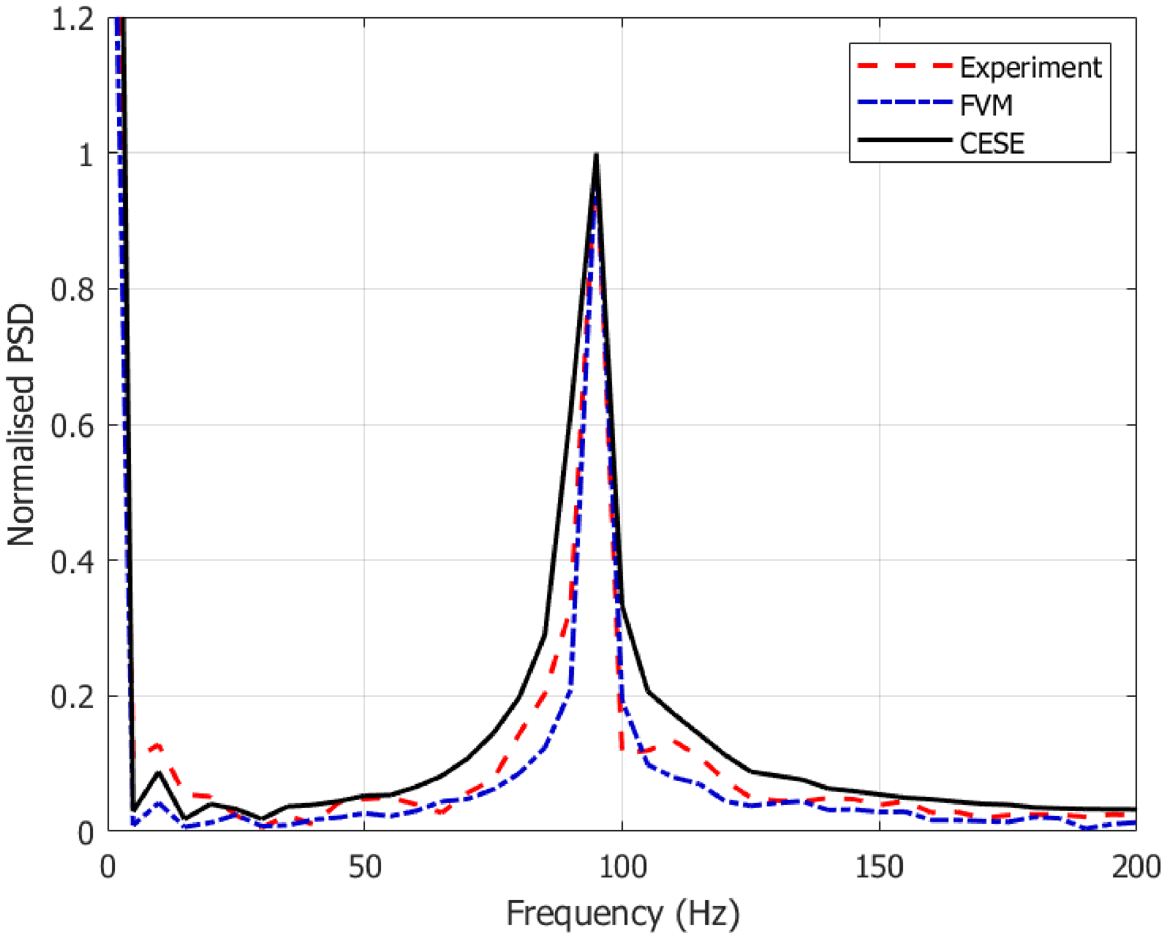


Length (mm)	Width (mm)	Thickness (mm)	Young's Modulus (GPa)	Density (kg/m^3)	Damping Ratio
130 (±0.1)	80 (±0.1)	1.95 (±0.1)	52.7 (±0.1)	2668 (±0.1)	0.0038

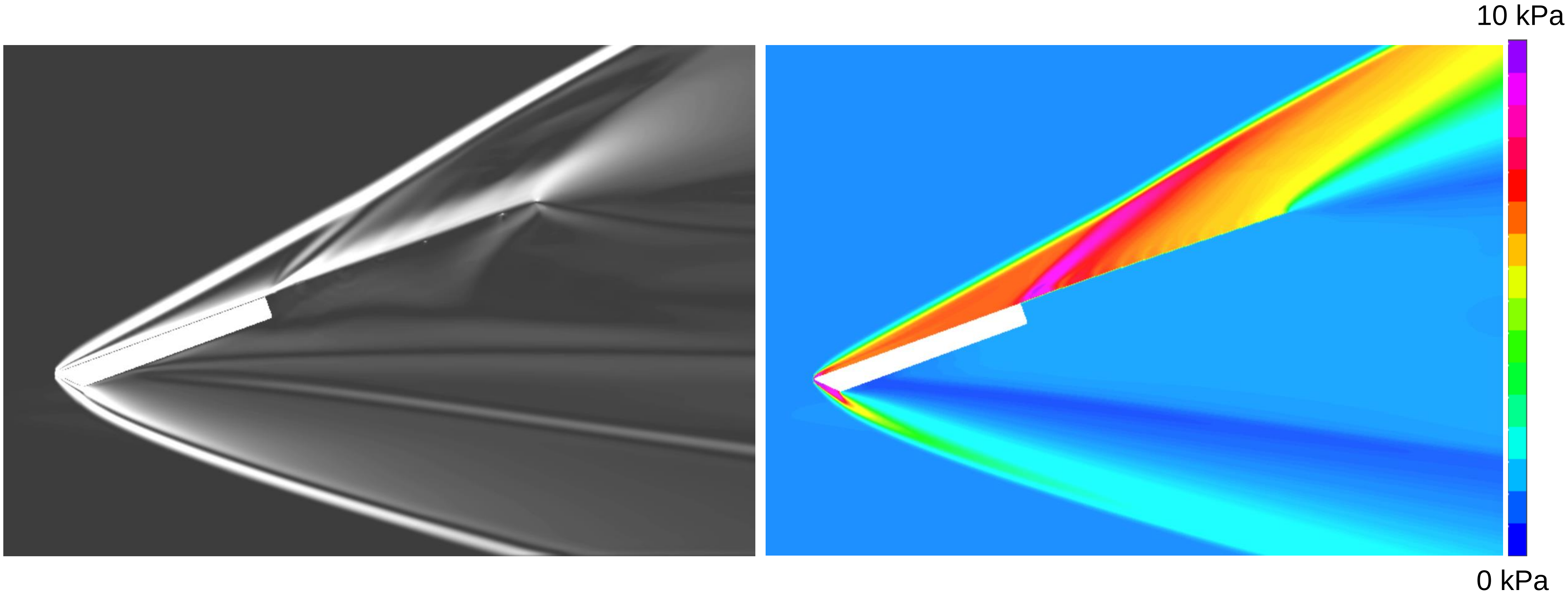
Freestream Mach	Freestream Pressure (Pa)	Freestream Temperature (K)	Freestream Density (kg/m^3)	Wall Temperature (K)	Reynold's Number (m^{-1})
5.85	755	75	0.0351	290	7.16×10^6

Preliminary Results





Preliminary Results



Conclusion

- Preliminary results are promising
- Parameters need to be refined to improve accuracy

- LS-DYNA is not supporting CESE as a CFD package
- There is a need to develop an in-house or open source CESE code
  - Easily parallelized means strong potential for GPU acceleration

Connect with me!