Time-efficient simulations of weapon bay in fighter aircraft

Karthick Rajkumar[1)], Eike Tangermann[1)], Markus Klein[1)]
Sebastian Ketterl[2)], Andreas Winkler[2)]

[1)]Bundeswehr Universität München,
Werner-Heisenberg-Weg 39, 85577 Neubiberg, Germany
[2)]Division of Flight Physics - Aerodynamics - Aerodynamic Design,
Airbus Defence and Space, 80577 Manching, Germany
karthick.rajkumar@unibw.de

A cavity flow can be seen in many real world applications. One such application is a weapon bay structure in fighter aircraft. A cavity configuration features a highly complex unsteady separated flow (see left of Fig.1), which is characterised by an intense aero-acoustic coupling mechanism. Strong pressure oscillations produced in and around the cavity have the potential to cause structural fatigue and induce resonance phenomena inside the cavity. It is of primary importance that this flow mechanism inside the cavity is understood and provide insights to control the relevant parameters.

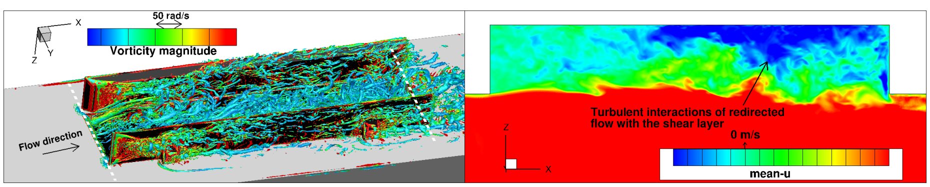

Fig. 1: Snapshots of instantaneous fields for Ma=1.2 using SA-IDDES model: Vortical structures in the cavity (left); streamwise velocity at an instant of time (right)

In this study, a open cavity configuration [1] with doors attached on the sides and a length to depth ratio of 5.7 has been studied numerically using the TAU code [2] developed by German Aerospace Center for high subsonic and supersonic flows. The flow is characterised by turbulent interactions between shear layer originating from the front edge of the cavity and the flow redirected off the downstream cavity wall (see right of Fig.1). Two turbulence resolving methods have been investigated in this study. The first method is based on the hybrid RANS-LES method using the SA-IDDES model [3] whereas the second method is based on scale-adaptive simulation using the SST-SAS model [4].

The study comprises the Mach numbers (Ma) 0.8 and 1.2 with Reynolds number (Re) 12×10^6. The Rossiter modes [5] occurring in the cavity due to the feedback mechanism have been numerically computed and validated for both flow conditions using reference data from experimental measurements. The SST-SAS model has found to be around 70 % computationally efficient compared to the hybrid RANS-LES model with reasonable accuracy in capturing the Rossiter modes (see Fig. 2). Additionally, some quantative results have been compared between the two scale-resolving methods and practical guidelines to simulate the cavity flows have been outlined.

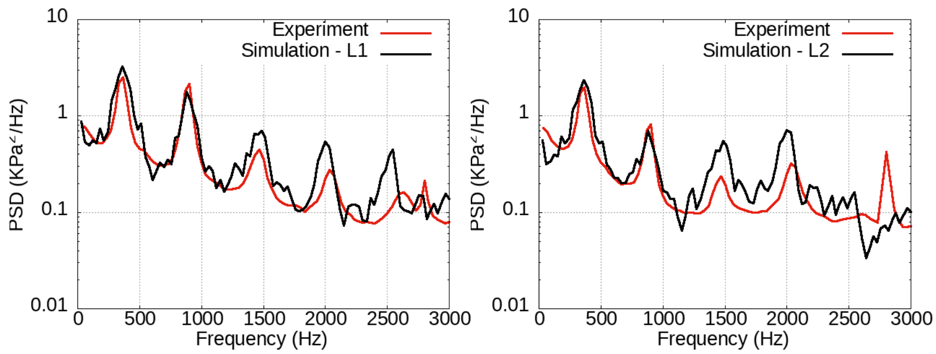

Fig. 2: Validation of PSD computed using SST-SAS model (Ma=1.2) at some of the probe locations along the ceiling.

Keywords: cavity flow, the Rossiter modes, hybrid RANS-LES methods, Scale-adaptive simulations(SAS)

References

1. Mayer, F., Mancini, S., Kolb, A.: Experimental Investigation of installation effects on the aeroacoustic behavior of rectangular cavities at high subsonic and supersonic speed, presented at Deutscher Luft- und Raumfahrtkongress, Germany (2020).
2. Langer, S., Schwppe, A., Kroll, N.: The DLR Flow Solver TAU - Status and Recent Algorithmic Developments. In: 52nd Aerospace Sciences Meeting, National Harbor, Maryland, USA (2014).
3. Spalart, P.R., Jou, W.H., Strelets, M., Allmaras, S.R.: Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach, First AFOSR International Conference on DNS/LES, Ruston, LA, in: Advances in DNS/LES, edited by C. Liu and Z. Liu, 4-8, August (1997).
4. Chaouat, B.: The State of the Art of Hybrid RANS/LES Modeling for the Simulation of Turbulent Flows. Flow Turbulence Combust 99:279327 (2017).
5. Rossiter, J.E.: Wind-Tunnel Experiments on the flow over rectangular cavities at Subsonic and transonic speeds. Report and Memoranda 3438, Aeronautical Research Council, London, UK (1966).