Substantial efficiency increase in gas turbines through direct use of coupled unsteady combustion and flow dynamics

Inflow of cofluent starting jets into a plenum with minimal losses

Prof. Dr. J. Sesterhenn, E. Büchner, *Institute for Computer Fluid Dynamics and Technical Acoustics, TU Berlin*

In Short

- importances of the investigation of the Pulsed detonation engine
- PDE unstedy process and shock losses
- geometry variation for finding a nozzle prototype

Pulsed detonation engine (PDE) is the future of the engine market. Instead of constant pressure combustion, this engine's function is based on constant volume combustion. In comparison to a conventional engine, the pulsed detonation engine has a higher efficiency [1]. Thus, the use of the engine is more affordable and sustainable. However, there are two major challenges that have not been solved so far. Due to the high-pressure differences between the combustion chamber and the outlet, strong shocks are expected to occur, and the shocks could significantly reduce the efficiency of the combustion process. Another challenge is that a weak suction wave of the pulsed detonation engine often leads to disorder of the combustion chamber. To overcome these challenges, a CFD simulation needs to be implemented with the aim to develop a clearer idea of the ideal geometry of the nozzle for a pulsed detonation engine, and subsequently build a prototype. Since direct numerical simulation would be too expensive to perform and Reynolds-averaged Navier?Stokes equations simulation has not sufficient resolution, a Large-Eddy simulation (LES) can be regarded as the most adequate alternative. Here, the composition of this project will be presented.

Common engines are designed for a constant mass flow out of the combustion chamber. An overview for designing the rocket nozzle for a stationary operating engine is given by [2] and [3]. Nevertheless, so far, not much research has been done on pulsed detonation engines with the main focus on wave work for different nozzle geometries. In case of a PDE, the combustion process is unsteady. After the detonation, the flow comes out of the combustion chamber and time-dependent phenomena take place, (figure 1).At the same time, when certain criteria are met, an expansion wave is traveling backward into the combustion chamber and is reflected at the combustion chamber wall (opposite the nozzle). We call this wave "suction wave". This suction wave is used to fill the combustion chamber with new fuel and fresh air. The principle is the same as in a manual air pump for bicycles. The piston is traveling out of the tube, creating underpressure, and fresh air flows in through the valve. For this pumping effect to be effective, the suction wave should be strong to support the filling process. The strong impact of shock losses is shown in figure 2. With a higher Mach-number, that means with a higher pressure difference between the combustion chamber and nozzle, the generated entropy increases linearly for a Mach number bigger than 2.

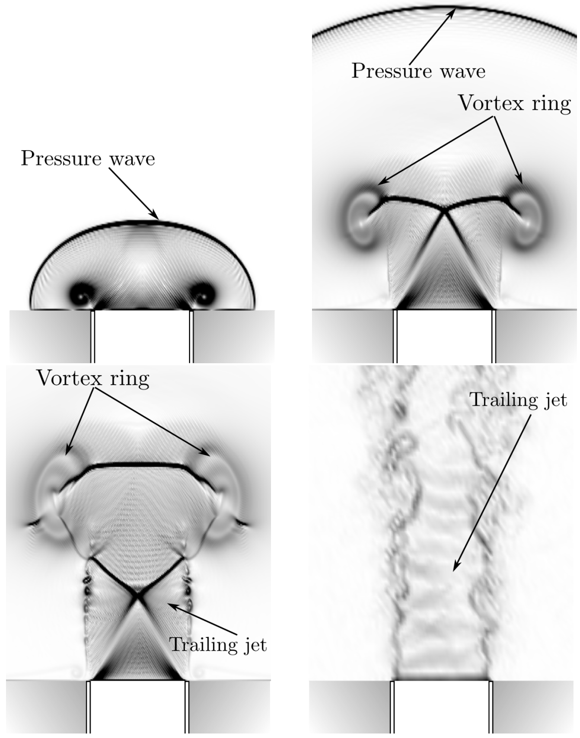

Figure 1: *numerical Schlieren from a three-dimensional simulation the combustion chamber outcoming flow, unsteady phenomena, [4]*

The aim of this project is to find a nozzle prototype for the given PDE setup to reduce entropy generation and to get a strong suction wave. The PDE setup is given by the initial condition for the inlet pressure of 4 bar and the outlet of 1 bar. The geometry will be varied in shape and length. The strategies to gain further insights into the possibilities to maximize efficiency and therefore minimize shock strength are to slow down the jet to get a lower Mach number

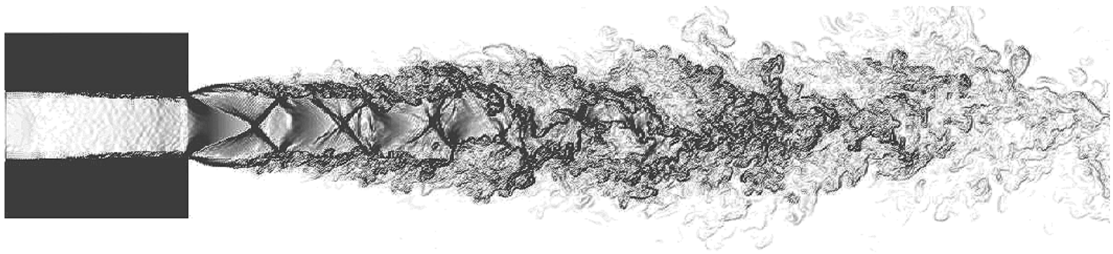

Figure 3: numerical Schlieren from a three-dimensional simulation the turbulent supersonic jet with a convergent nozzle. Geometry was realized by using the volume penalization method, [5]

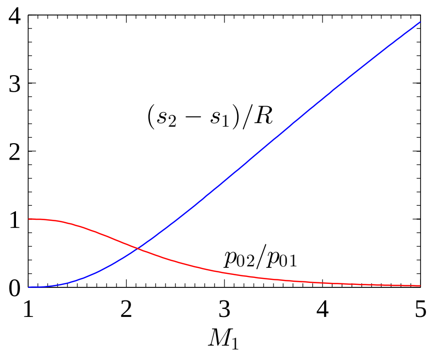

Figure 2: shock losses from a straight shock as a function of the shock Mach number (blue), total pressure losses (red)

(see figure 2) or to create a turbulence for decreasing the time average velocity The complex shape will be realized with the volume penalization method, figure 3. In the computational domain, the entropy will be balanced in order to see the difference in output between the various nozzle designs. In addition, at the nozzle inlet, the pressure over the time will be measured and strength of the suction wave evaluated.

WWW

http://www.sfb1029.tu-berlin.de/menue/projects/c02/parameter/en/

More Information

[1] Kailasanath, K., Recent developments in the research on pulse detonation engines Recent Developments in the Research on Pulse Detonation Engines. AIAA Journal, 41(2):145–159, 2003.

[2] Östlund, J., Flow Processes in rocket engine nozzles with focus on flow separation and side-loads. Dissertation, KTH Royal Institute of Technology, 2009.

[3] Hagemann, G. and M.Frey, Shock pattern in the plume of rocket nozzles: needs for design considerationShock pattern in the plume of rocket nozzles: needs for design consideration. Shock Waves, 17(6):387–395, 2008.

[4] Juan José Peña Fernández, The compressible starting jet: fluid mechanics and acoustics, Dissertation, Berlin 2017

[5] Schulze, J., Adjoint based jet-noise minimation. Dissertation, Technische Universität Berlin, 2013

Funding

CRC 1029 (Project C02)