

Simulation (CFD/FEM)

Experiment

Construction

Theory

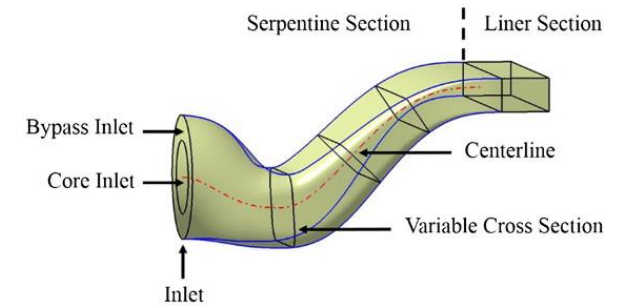
Numerical investigation of a curved outlet duct for small gas turbines

Overview:

At the Institute of Jet Propulsion (ISA) of the University of the German Armed Forces Munich (UniBwM), novel concepts for thrust-vector manipulation of subsonic aircraft and their integration into the overall system are investigated. Concepts for thrust-vector sweeping can lead to an increase in agility and/or a reduction of the so-called radar cross-section and infrared signature. Due to flight system dynamic conditions, the jet engine and the associated nozzle cannot always be designed to be coaxial with each other. In these cases, the turbine exit mass flow must be routed to the nozzle via a curved exit duct. Within the scope of this work, such an outlet duct is to be designed and numerically (CFD) investigated. Special focus will be put on the effects of the geometric parameters on secondary flows and pressure losses in the outlet duct.

Objectives:

- Design (analytical), construction (CAD) and detailed numerical investigation (CFD) of a curved outlet channel
- Identification of influential parameters (geometric and aerodynamic) on the flow behavior in the outlet duct



Sketch of double serpentine nozzle

Requirements:

- Interest in numerical flow simulation and CAD (ideally, basic theoretical knowledge and first practical experience with flow solvers exist)
- Interest in fluid mechanics and the application of learned theory
- Independent and target-oriented way of working

Begin: Immediately

Contact: Nils Schwagerus

E-Mail: nils.schwagerus@unibw.de

Telephone: 089 6004 2216

Office: Buil. 37/200, Room 1209a