Title: Reduced order modeling of some fluid flows of industrial interest

Author: J. M. Vega (UPM,ETSI Aeronáuticos Madrid, Spain)

Instabilities and bifurcations are common in fluid flows of industrial interest and thus they will be (sooner or later) unavoidably relevant in design and certification of industrial devices.
Instabilities may have a detrimental effect on either safety (e.g., flutter instability may produce large vibrations in aircrafts) or performances (e.g., laminar/turbulent transition increases skin friction in boundary layers) but they can also be beneficial (to, e.g., promote convection in micro-heat-exchangers). In the latter case, the required analysis should include not only the instability limits but also the whole (or at least a part of the) bifurcation diagram, which usually requires a huge computational effort, not compatible with the strict cost and time constrains that are common in industry. For instance, aerodynamic calculations in conceptual design of commercial aircrafts is currently based on either semi-empirical formulae or quite low fidelity models, which must be tested/corrected by experimental tests.

Reduced order models (ROMs) may help to fill this gap. A class of ROMs is based on the projection of the governing equations onto a low dimensional manifold. This can be determined applying proper orthogonal decomposition (POD) to a set of previously calculated snapshots that should be representative of the considered flow patterns. The snapshots can be obtained using a computational fluid dynamics (CFD) solver on the full set of governing equations. The method can be applied to both steady and time-dependent problems. Obtaining robust, computationally efficient ROMs require to attend several difficulties/opportunities, in connection with, e.g.:

- The inner product that is used to project the governing equations, which can be based on the whole computational domain, but also on only a limited number of mesh points.

- The equations that are actually projected, which can be either the exact governing equations or the equations that are actually used by the CFD solver (numerical artifacts included).

- The use of industrial/open source CFD solvers, whose details are not known.

- The role of CFD errors.

For illustration, several examples on micro-fluidics and transonic aerodynamics will be considered.