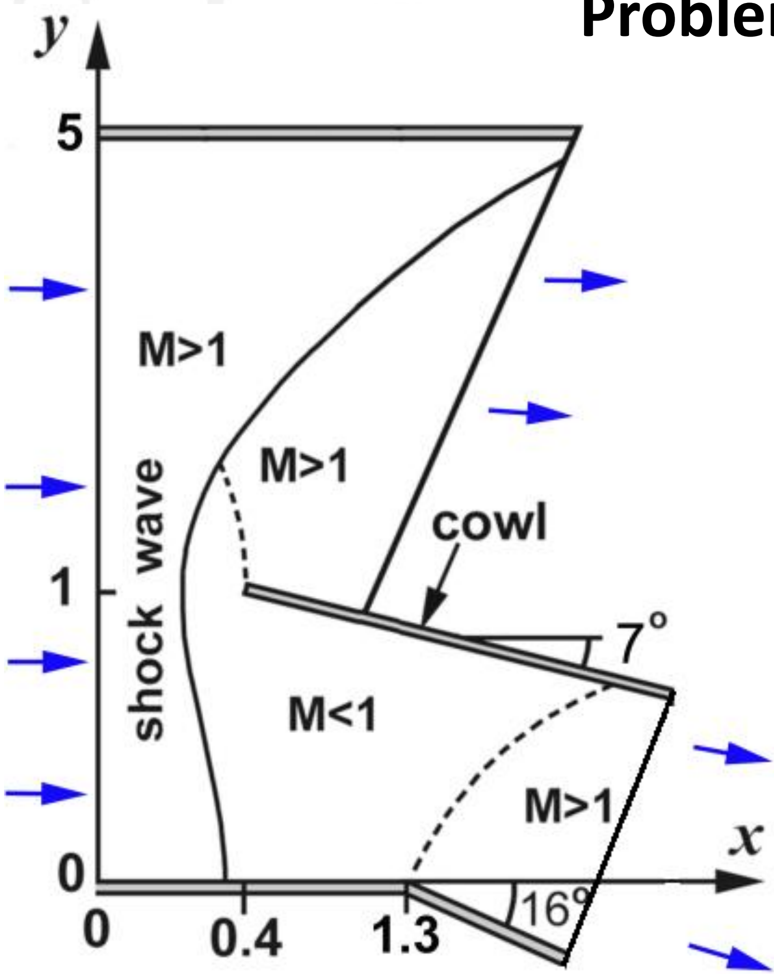
Санкт-Петербургский
государственный
университет
www.spbu.ru

TRANSONIC FLOW SIMULATION IN A BENT CHANNEL USING SU2 SOFTWARE

Anatoly Ryabinin, Alexander Kuzmin

Department of Hydroaeromechanics
Saint Petersburg University
Saint Petersburg, Russia
a.ryabinin@spbu.ru a.kuzmin@spbu.ru

Problem Formulation



Channel 1

Lower wall:

$y(x) = 0$ at $0 < x < 1.3$;

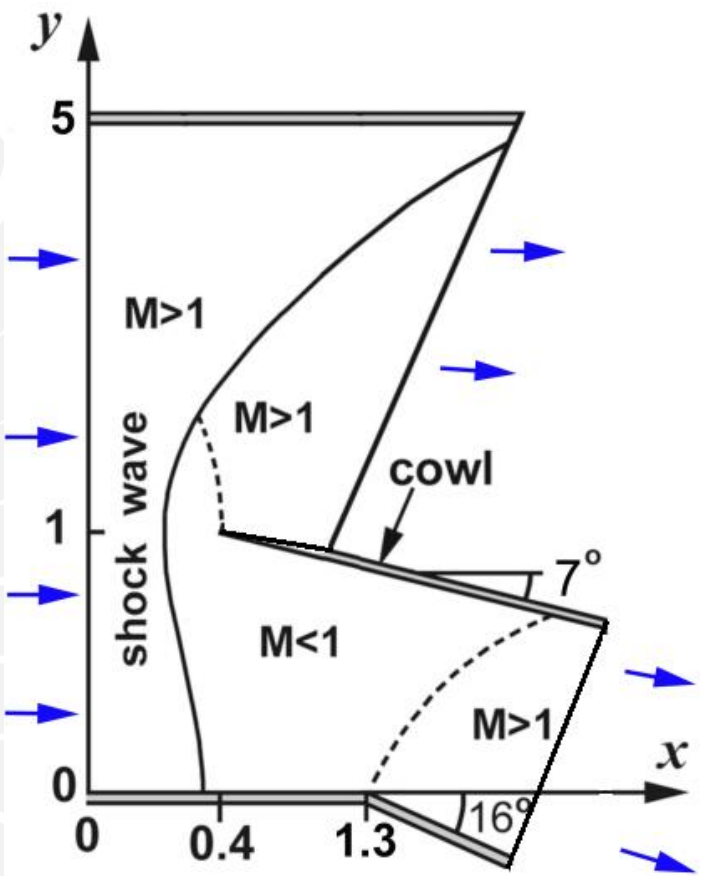
$y(x) = -(x - 1.3) \tan 16^o$ at $1.3 < x < 1.5$.

Cowl:

$y(x) = 1 - (x - 0.4) \tan 7^o$ at $0.4 < x < 2.0$

Channel 2

M_{in}
p_{in}
T_{in}
$\alpha=0$



Mesh Generation

The meshes were generated with package Gmsh
(Ch. Geuzaine, J.-F. Remacle, Gmsh Reference Manual, 278 p., 2014.)
URL: http://geuz.org/gmsh/doc/texinfo/gmsh.pdf.

Computational meshes were constituted by quadrangles in 35 layers on the walls and by triangles in the remaining region.

The total number of mesh cells was about 2×10^5. Some test calculations were performed with the doubled number of mesh elements. We also varied the thickness of the lower layer of the mesh.

The SU2 2D mesh was transformed into a 3D mesh, whose lateral size was equal to one element. The transformed mesh is in the TGrid / Fluent format which is suitable for the calculation in the package Ansys CFX.

SU2 details

To solve URANS equations, we used Stanford University Unstructured (SU2) release 7.0.0 “Blackbird”.

The averaged Courant-Friedricks-Levy (CFL) number of the grid was less than 3.

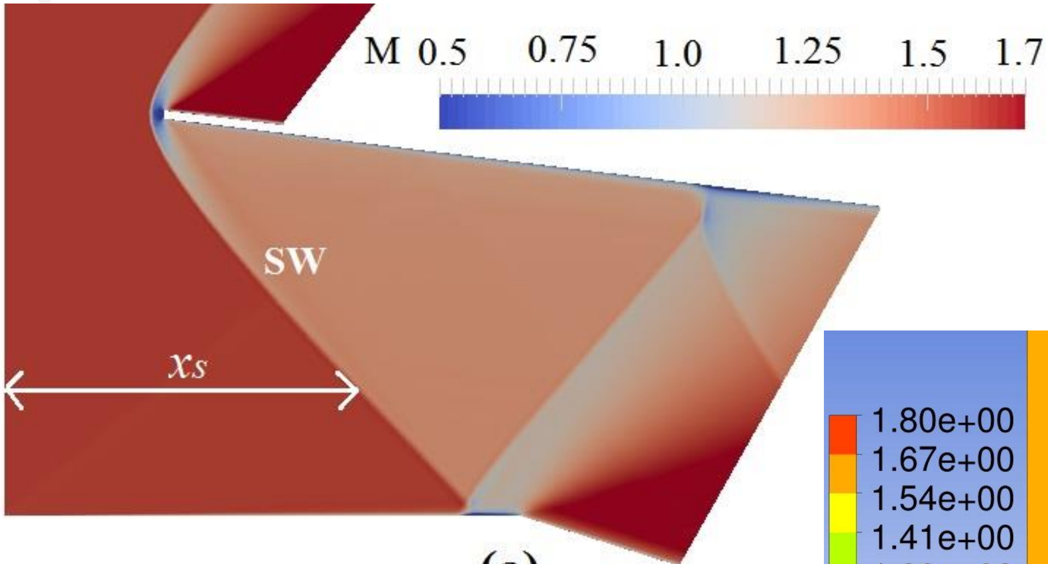
The FGMRES algorithm proposed by Saad was chosen for the linear solver for implicit formulation.[Y. Saad, SIAM J. Sci. Comput. Vol. 14, pp. 461-469, 1993.]

Minimum error of the linear solver for the implicit formulation was equal to 10^{-10}.

Jameson-Schmidt-Turkel (JST) scheme was used for convective terms. [A. Jameson, W. Schmidt and E. Turkel, AIAA paper 81-1259, 1981]

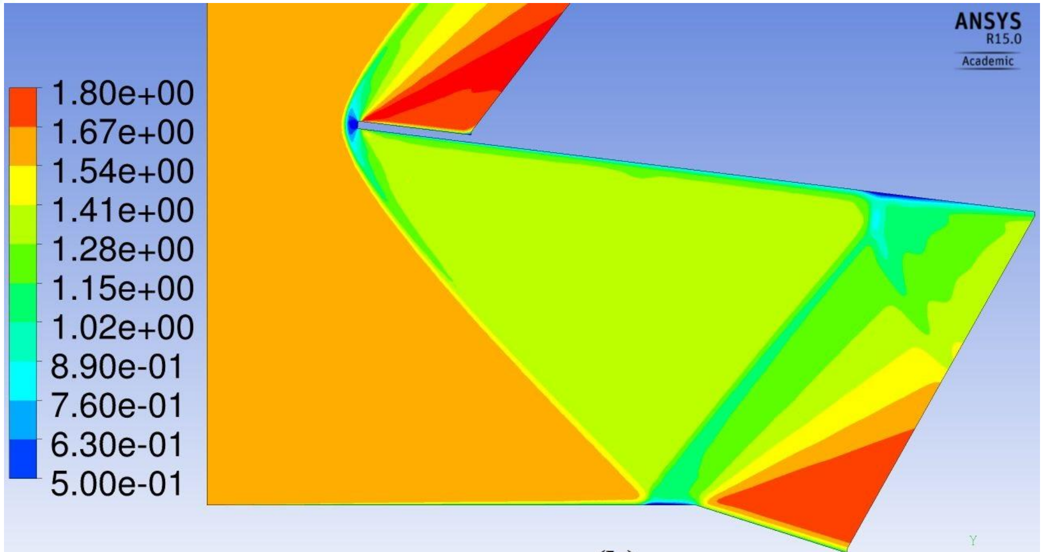
We employed the slope limiter proposed by Venkatakrishnan.

[V. Venkatakrishnan, AIAA paper 93-06680. 1993].



(a)

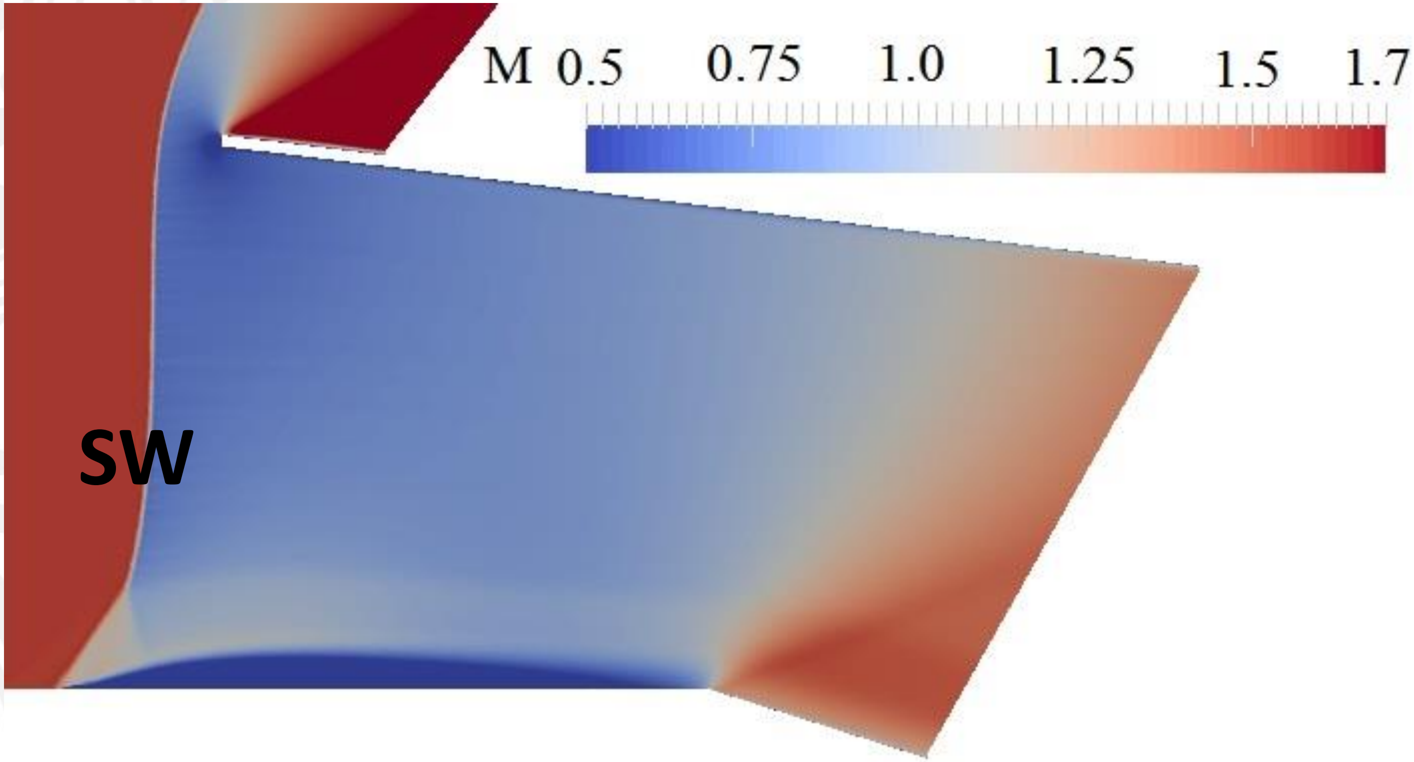
Computations with SU2



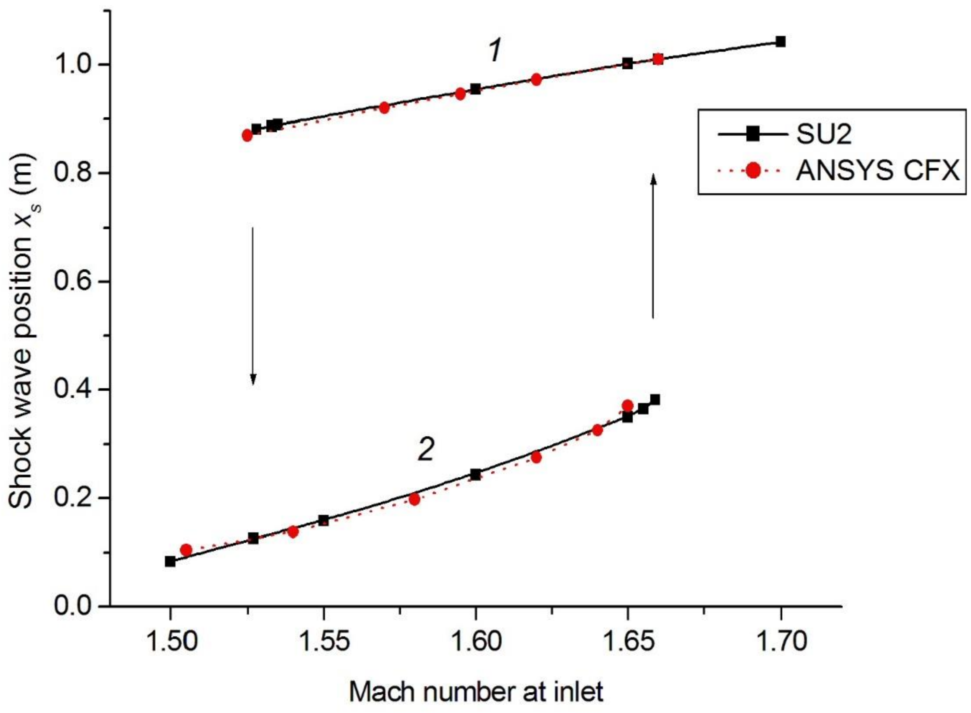
(b)

Computations with ANSYS CFX-15

Mach number contours in channel 1
at M_{in} = 1.6, Re= 2.2×10^{7},
the flow regime with swallowed shocks,
Spalart-Allmaras turbulence model



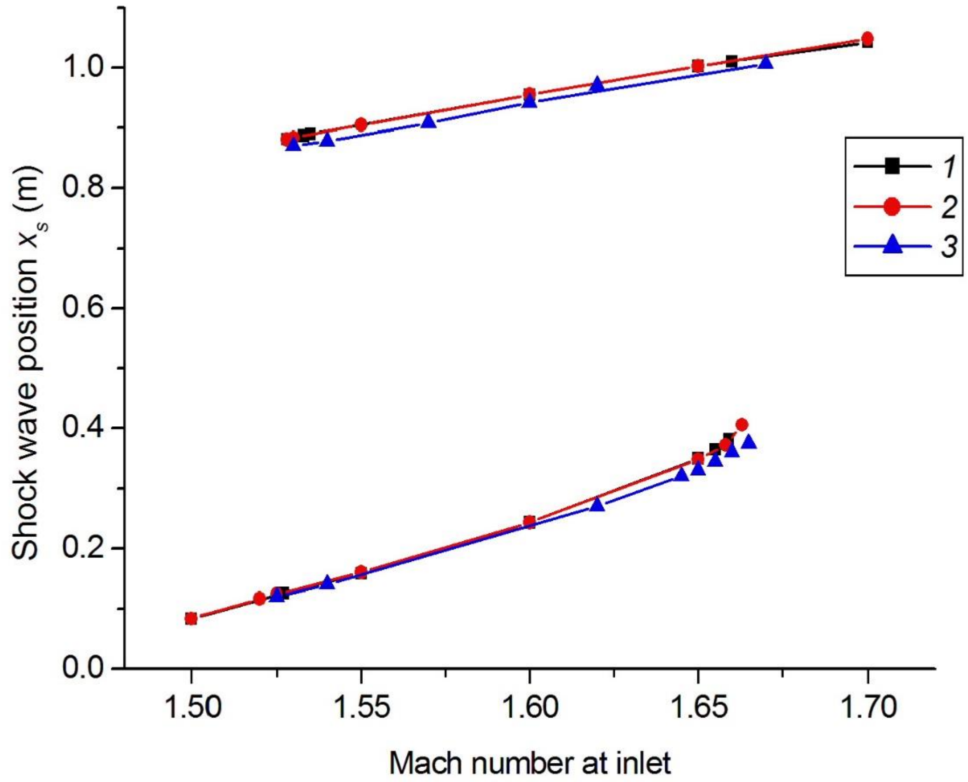
Mach number contours in channel 1 at M_{in} = 1.6, Re= 2.2×10^7
The flow regime with the expelled shock SW. Computations with SU2.



Coordinate x_s of the shock wave SW versus Mach number at inlet for channel 1, Re= 2.2×10^7, p_{in} = 50000 Pa, Spalart-Allmaras.

Solid curves: computations with SU2;

dashed curves: computations with ANSYS CFX.

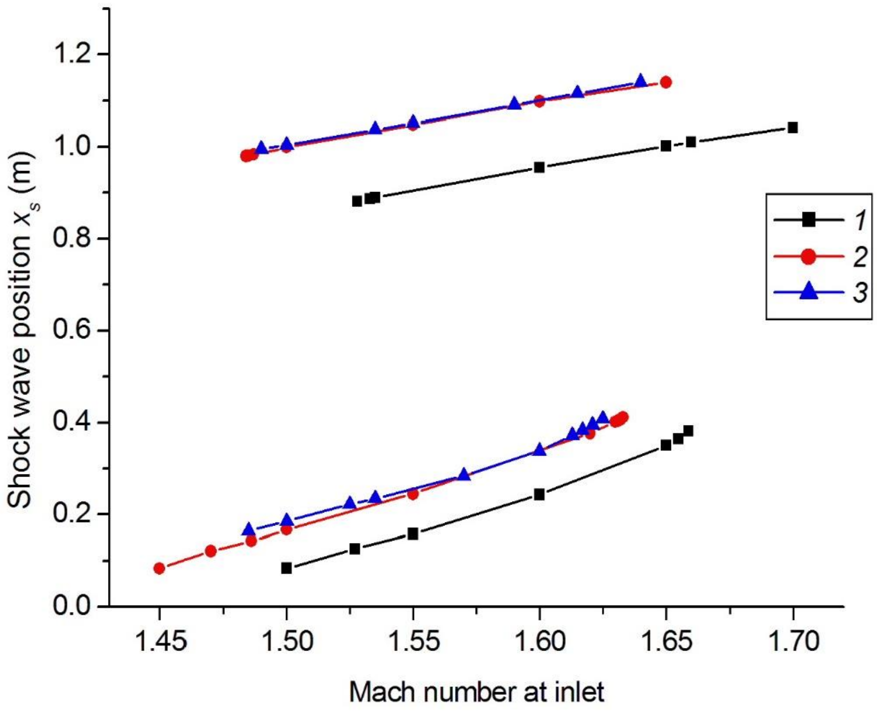


Coordinate x_s of the shock wave SW versus Mach number at inlet for channel 1.

1 — SU2, p_{in} = 50000 Pa, Spalart-Allmaras turbulence model;

2 — SU2, p_{in} = 25000 Pa, Spalart-Allmaras turbulence model;

3 — Ansys CFX, p_{in} = 50000 Pa, k - ω SST turbulence model.



Coordinate x_s of the shock wave SW versus Mach number at inlet for channels 1 and 2, p_{in} = 50000 Pa.

1 — SU2, channel 1; 2 — SU2, channel 2; 3— Ansys CFX, channel 2,

Conclusions

- Calculations of the transonic flow in a bent channel using a freely distributed SU2 package revealed the existence of a hysteresis in a certain range of Mach numbers given at the inlet of the channel.
- The boundaries of the hysteresis range are weakly dependent on the Reynolds number in the range from 1.1×10^7 to 2.2×10^7.
- <u>A comparison of the solutions obtained using SU2 and Ansys CFX with Spalart-Allmaras and k - ω SST turbulence models showed that the solutions are close to each other.</u>
- The boundaries of the hysteresis range for channel with sharp edge of the cowl are shifted to the smaller inflow Mach number in comparison to the channel with blunt edge of the cowl.

Thank you for your attention!