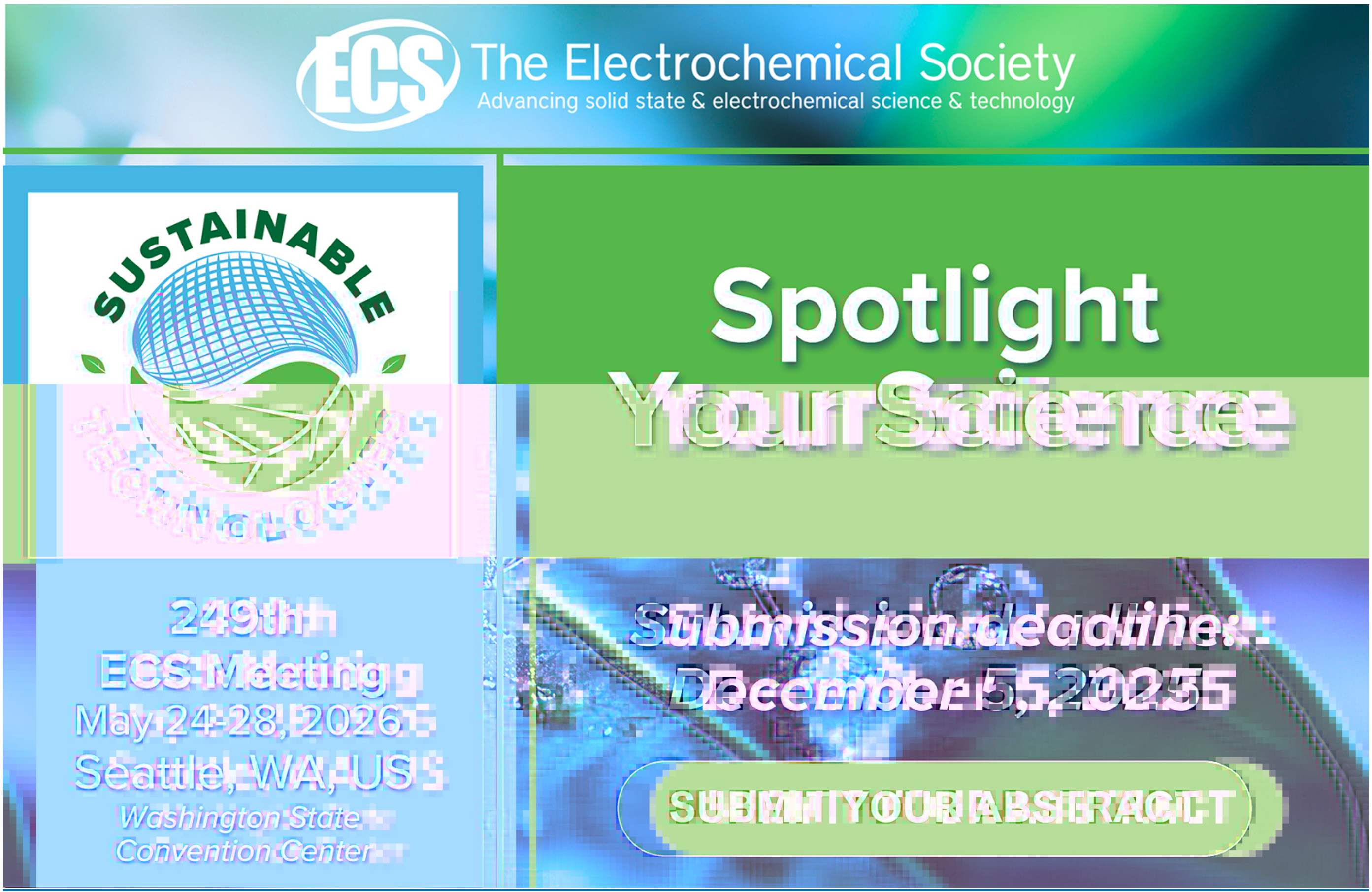
IOP Conference Series:
Materials Science and
Engineering

PURPOSE-LED
PUBLISHING™

PAPER • **OPEN ACCESS**

Investigations of pressure field along a channel of a pressure wave supercharger

To cite this article: I Costiuc *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1220** 012023



Investigations of pressure field along a channel of a pressure wave supercharger

I Costiuc[1], L Costiuc[2], A Chiru[1]

[1]Automotive and Transport Engineering Department, Faculty of Mechanical Engineering, Transilvania University of Brașov, BV 500036, Romania
[2]Mechanical Engineering Department, Faculty of Mechanical Engineering, Transilvania University of Brașov, BV 500036, Romania

Abstract. The aim of the paper is a numerical investigation of the evolution of the pressure field along the wave rotor channels of a pressure wave ICE supercharger. In the present literature, most of the studies are considering the fluids as incompressible and inviscid in a 2D field. The present study is using the compressible and viscous terms in the unsteady Lattice Boltzmann method for fluid in a 3D field. The geometry was drawn in CAD software using measurements made on a real model of the CX-93 pressure wave supercharger. The simulation was conducted using a code for native unsteady LBM approach to reproduce data such as pressures, temperature and mass flows, which are usually hard to be measured in a real pressure wave supercharger. The computational domain was modelled as a moving rotational domain with adaptive refinement. Results such as velocity, pressure and temperature field in the rotor channels were obtained for exhaust gas inlet pressure of 0.292 MPa and 721 K temperature at different rotational speeds. The air inlet state considered was: 0,096 MPa and 313 K. The simulated values obtained are similar to the reported experimental results found in the literature showing a good concordance with the model.

1. Introduction

A supercharger is a device positioned on the intake side of the engine to increase the air pressure at the inlet to the engine cylinder. By using a mechanical supercharger, the improvement of the thermal efficiency of the internal combustion engine (ICE) is limited, because part of the effective power of the ICE is consumed to drive the compressor, as shown in [1, 2]. Another conventional approach to increase the fresh air pressure is the use of the exhaust gas energy within turbines, which drive a suction air compressor (turbocharging). Turbochargers are the most widely used solutions for car manufacturers to produce useful boost, because of their advantages, such as a higher thermal efficiency due to the energy recovery of the engine’s exhaust gas.

Other constructive alternatives for supercharging manage to overcome the shortcomings of the engine turbocharger, such as the turbo-lag [2]. One alternative is the pressure wave supercharger (PWS). Pressure wave aggregates, known also as wave rotors, use shock waves to transfer energy directly between fluids without using mechanical components, thus having the potential to increase efficiency [3, 5-7]. In a PWS, there is a direct interaction between high pressure and high temperature exhaust gas and low pressure and low temperature air. In short, hot gases produce a shock wave that extends through the wave rotor channels and compresses the fresh air. The quick response to engine torque for the full range of engine speeds and the intake air pressure boost are reasons to consider PWS a good ICE supercharging option for vehicles.

The present paper aim is a numerical investigation of the evolution of pressure field along the wave rotor channels for a PWS, taking into account the compressible and viscous character of the fluids when using the unsteady Lattice Boltzmann method for fluids in a 3D field [7]. The PWS was modelled in CAD software after the real geometry of a COMPREX CX-93 pressure wave supercharger.

2. Numerical model

The numerical modelling used a 3D CFD model using FEM for PWS [4]. The 3D approach showed the distribution of the velocity, pressure and temperature field in the entire modelled volume. This approach involves a consistent calculation effort, even if the fluid is considered inviscid or in a stationary state. That is due to a large number of finite elements used. In the present study, it is used a 3D numerical model using the Lattice Boltzmann Method. This method was chosen because it is implicitly non-stationary and allows native modelling of rotational motion. The implementation of the LBM method in XFlow-2013 made possible the realistic simulation of the possible conditions in the PWS channels. The 3D-PWS channels in the rotor are modelled using the basic dimensions for the CX-93 pressure wave rotor; also, the dimensions of the cold and hot side of the PWS with compression and expansion pockets are considered. The 3D geometry was created using AutoCAD as shown in figure 1 and figure 2.

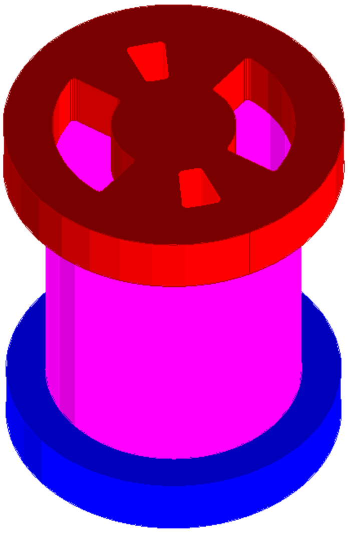


Figure 1. PWS cold air (blue) and hot gases (red) side CAD geometry.

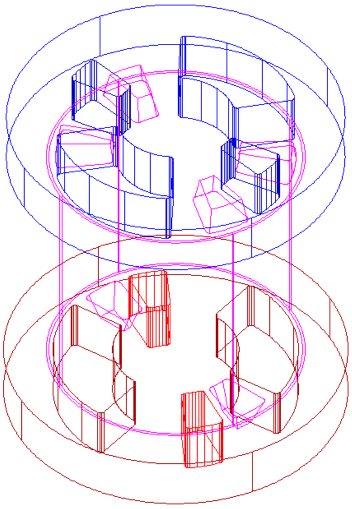


Figure 2. PWS raster geometry.

The final numerical model, after some import resulting geometry repair operations, is shown in figure 3 and figure 4 and consists of 42839 D3Q27 elements. The rotor material used in modelling was considered steel and the fluid used is air. The fluid was considered a compressible gas, and the specific heat, thermal conductivity and viscosity were considered as temperature dependent.

An implicit solver model was used, coupling the conservation and momentum equations with the energy equation and the flow was treated as turbulent and time dependent. To simulate the PWS behaviour in the rotating machinery model a fixed rotational speed of 12500min^{-1} was used. The simulation time was set to 10 seconds to investigate the PWS stable response.

The boundary conditions at inlet and outlet ports were set up as: a pressure, temperature input on the hot side of the inlet gases duct, and a pressure and temperature input at the cold side for fresh air inlet duct. The PWS channels and the ports were initially assumed to be filled with air at a reference constant pressure and temperature. The entire rotating domain containing fluid and an axial motion was set up with the rotational speed.

To investigate the pressure in channels, a plot variable line is used as it is shown in figure 3, where the rotor is hidden, and figure 4, where the hot side is hidden. This plot line setting connects the inlet of hot gases with the exit of the air and is used to visualize the pressure distribution.

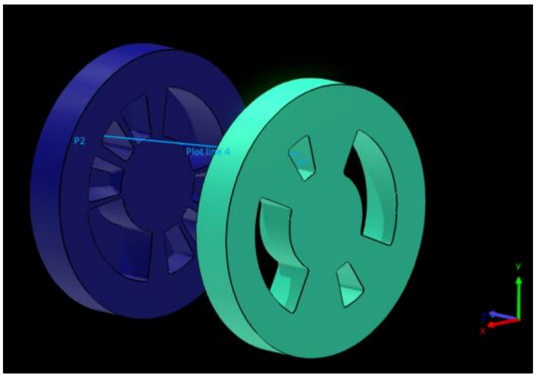


Figure 3. PWS cold air (blue) and hot gases (green) side CAD geometry.

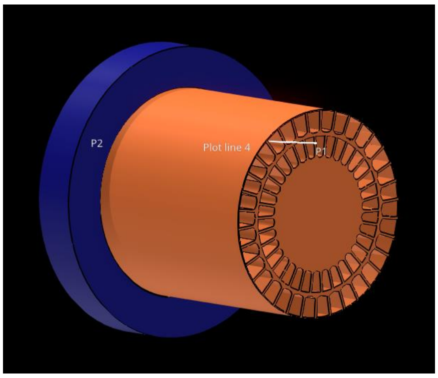


Figure 4. PWS rotor geometry.

The model was solved using single phase fluid, resolving both momentum and energy equations, with an enabled viscous term in the energy equation. The equation of state for air is used as Boussinesq state equation model. For viscosity, the Sutherland model is used. The solution used for turbulence modelling is the Large Eddy Simulation (LES) with introducing an additional viscosity, called turbulent eddy viscosity to model the subgrid turbulence. The LES scheme uses the Wall-Adapting Local Eddy viscosity model as presented in that provides a consistent local eddy-viscosity and near wall behaviour. The near wall treatment in the boundary layer uses a generalized law of the wall that takes into account the effect of adverse and favourable pressure gradients as presented in [7].

The boundary conditions used in the simulation are presented in table 1.

Table 1. Entry data for the model

Property	Value	UM
Static cold-side inlet pressure ($P_{1,0}$)	$0.96 \cdot 10^5$	Nm^{-2}
Static cold-side inlet temperature ($T_{1,0}$)	313	K
Static hot-side inlet pressure ($P_{2,0}$)	$2.92 \cdot 10^5$	Nm^{-2}
Static hot-side inlet temperature ($T_{2,0}$)	721	K
Air specific heat ratio (k_{air})	1.4	-
Air specific gas constant (R_{air})	287	$Jkg^{-1}K^{-1}$
Channel length (L)	93	mm
Rotor diameter (D)	93	mm

The software code for the LBM model imposes a stability parameter against time [8] with values between 0 and 1, with respect the feedback regarding the stability of the numerical scheme as the Courant–Friedrichs–Lewy (CFL) condition. A stability parameter near to 1 means the stability of the numerical scheme is not ensured and the simulation may diverge. The ideal value should be around 0.2 - 0.4. In this simulation, the resulted stability parameter is between 0.18 and 0.31 as is presented in figure 5, captured at $3x10^{-3}$ seconds.

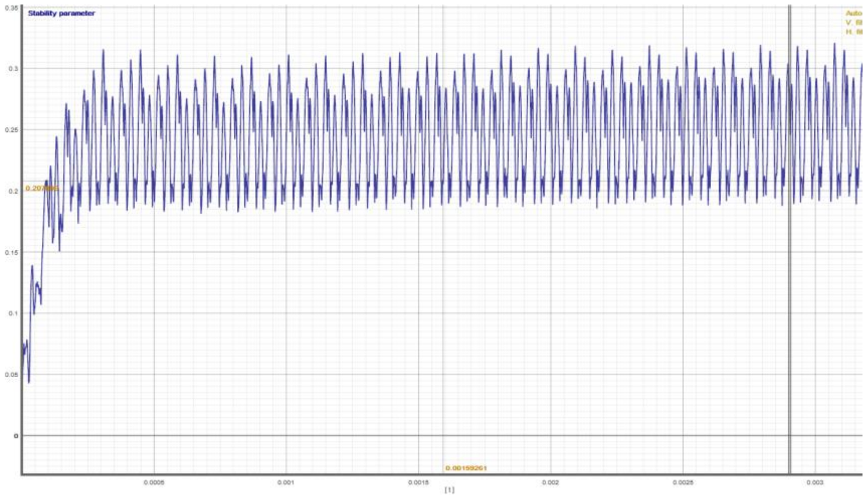


Figure 5. Stability parameter against time.

3. Results and discussions

The results obtained with the LBM model described above are presented below for the 19x10^{-3} seconds of simulation. The velocity distribution in PWS channels in the contour plot is presented in figure 6. The temperature field is presented in figure 7. The pressure field and the pressure distribution for a single PWS channel, connecting the exhaust gas inlet and the compressed air outlet, are presented in figure 8 and figure 9.

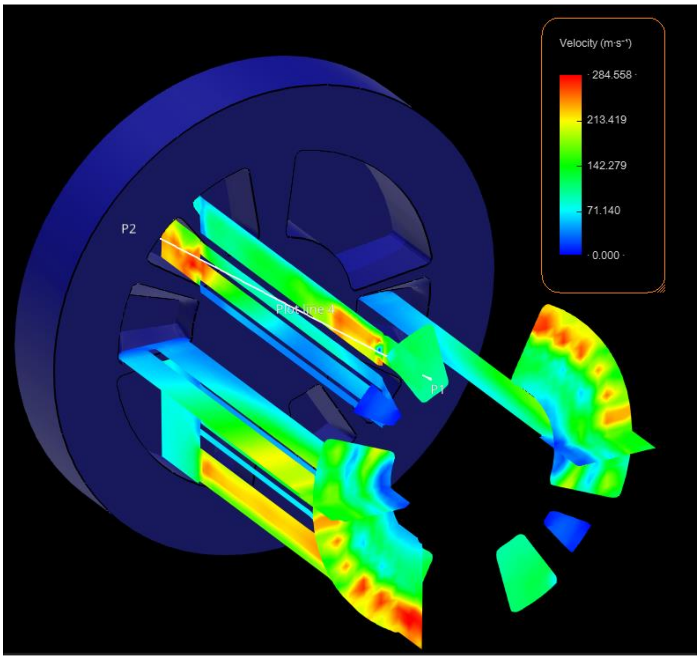


Figure 6. Velocity field for a 3D model using 3 cut-planes (XY, YZ and XZ).

From the simulation point of view, the model shows the non-uniform thermal and velocity fields. The velocity field, figure 6, shows that the variations of velocity in channels, as well in the output exhaust gas ports and output compressed ports were quite large from 70 to 284 m/s, depending on the position of the rotor channels.

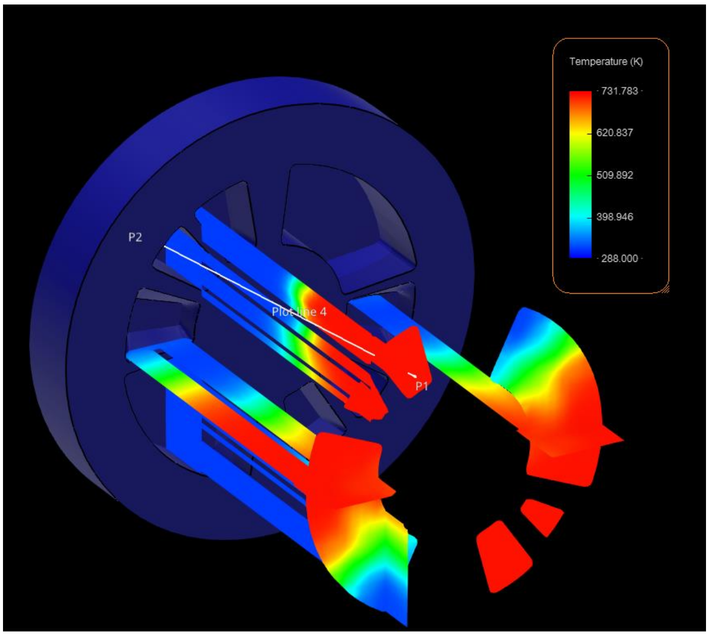


Figure 7. Temperature field for a 3D model using 3 cut-planes (XY, YZ and XZ).

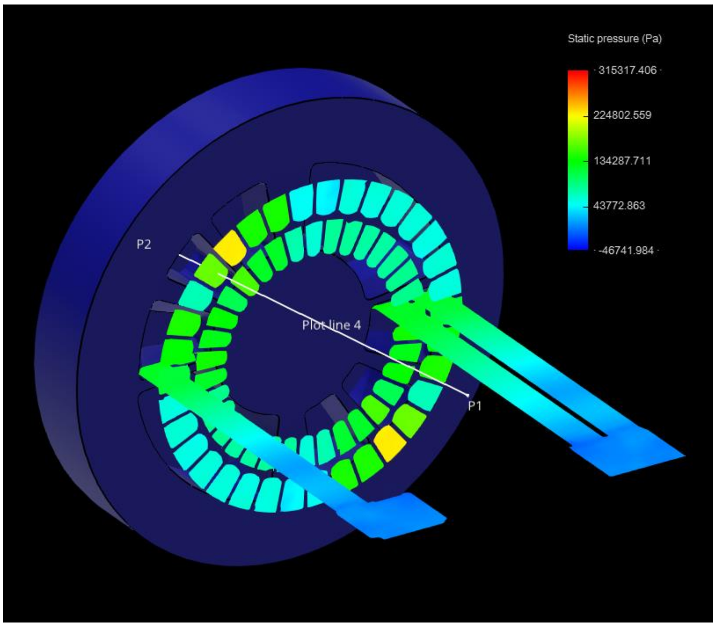


Figure 8. Pressure field for a 3D model using 3 cut-planes (XY, YZ and XZ).

The temperature field, as can be seen in figure 7, is also non-uniform along the rotor case. The main result for the temperature field is the fact that the interfaces of air-exhaust gases do not reach the cold side, and it remains almost near to the hot side (exhaust gases). The pressure field is presented in figure 8.

The pressure evolution along the channel is presented in figure 9 and shows that the compressed air is evacuated from PWS with a value of 1.47 bar, which correspond to a 1.53 compression ratio. These results are similar to the experimental values found in the numerous reported studies about CX-93 found in the literature, validating thus the present model.

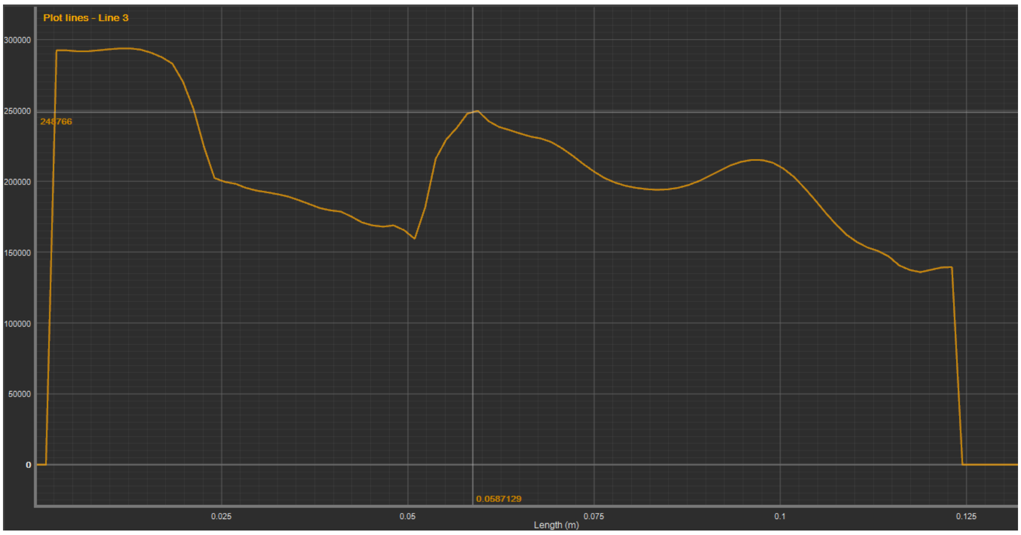


Figure 9. Pressure along a channel.

4. Conclusion

Using a wave rotor for the ICE supercharging reduces the baseline compression pressure ratio and the compression exit temperature. Furthermore, this may reduce the rotational speed which results in reduced mechanical and thermal stresses and relaxed design constraints.

The numerical results presented here encourage the idea that at microscale compression by shock waves may be more efficient than by conventional centrifugal compressors or turbo-compressors. However, the simulation results will be useful for further validation of the presented model with data considering variable rotational speed and experimental data. Also, a similar model for other different PWS types (dimensions, geometry, material, number of channels, etc) can be developed in future studies, as an instrument to improve any feature or performance of existing wave rotors.

References

[1] Spring P 2006 Modelling and Control of Pressure-Wave Supercharged Engine Systems, Diss. ETH No. 16490, Swiss Federal Institute of Technology, ETH Zurich

[2] Fu J, Liu J, Wang Y, Deng B, Yang Y, Feng R, Yang J 2014 A comparative study on various turbocharging approaches based on IC engine exhaust gas energy recovery, Appl. En.**113**

[3] Liu JP, Fu JQ, Ren CQ, Wang LJ, Xu ZX, Deng BL 2013 Comparison and analysis of engine exhaust gas energy recovery potential through various bottom cycles. Appl. Therm. Eng. **50**

[4] Costiuc I, Costiuc L 2017 Mesh based modeling of a pressure wave supercharger, Proc. Int. Conf. on Computational Mechanics and Virtual Engineering, COMEC 2017

[5] Iancu F, Piechna J, Müller N 2008 Basic design scheme for wave rotors, Shock Waves **18**, Springer Verlag

[6] Frackowiak M, Iancu F, Potrzebowski A, Ackbari P, Müller N, Piechna J 2004 Numerical Simulation of Unsteady - Flow Processes in Wave Rotors, Proc. of IMECE04 2004 ASME International Mechanical Engineering Congress, Anaheim, California USA

[7] Holman MD, Brionnaud RM, and Abiza Z 2012 Solution to industry benchmark problems with the Lattice-Boltzmann code Xflow, EU Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2012), J. Eberhardsteiner et.al. (eds.), Austria

[8] XFlow Tutorial Guide 2013. Next Limit, www.nextlimit.com