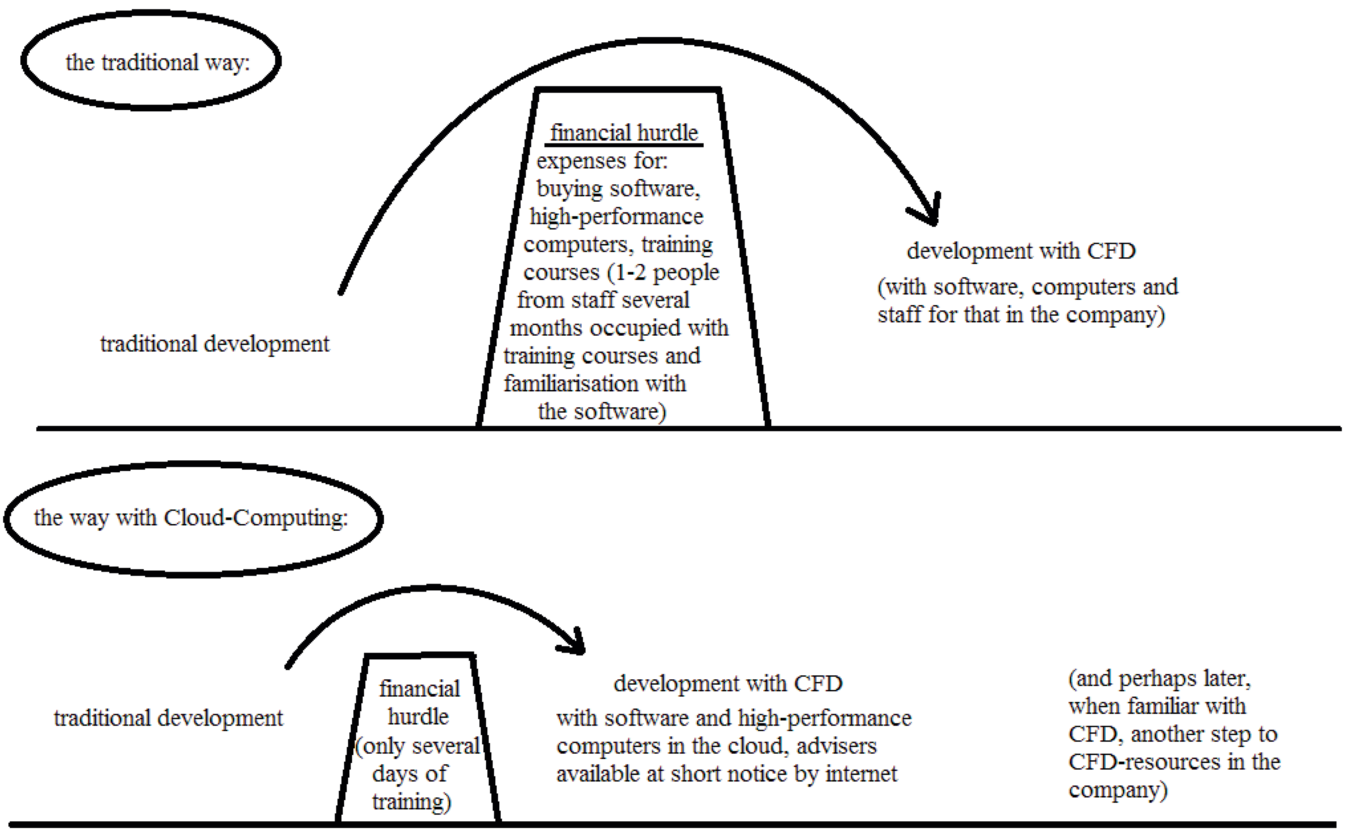
A Compendium about Cloud Computing in SME – the practical experience with "cloudy" CFD (computational flud dynamics -> calculating a flow instead of trial and error) in a medium-size enterprise



In the last years „cloud computing" was a buzz-word in industry. Paying for the use of computer resources and programs instead of buying them seems to be a good idea in many cases. In our case, the Boge company, medium-sized manufacturer and designer of industrial air compressors, we have been thinking about own CFD (computational flud dynamics -> calculating a flow instead of trial and error) for a longer time.

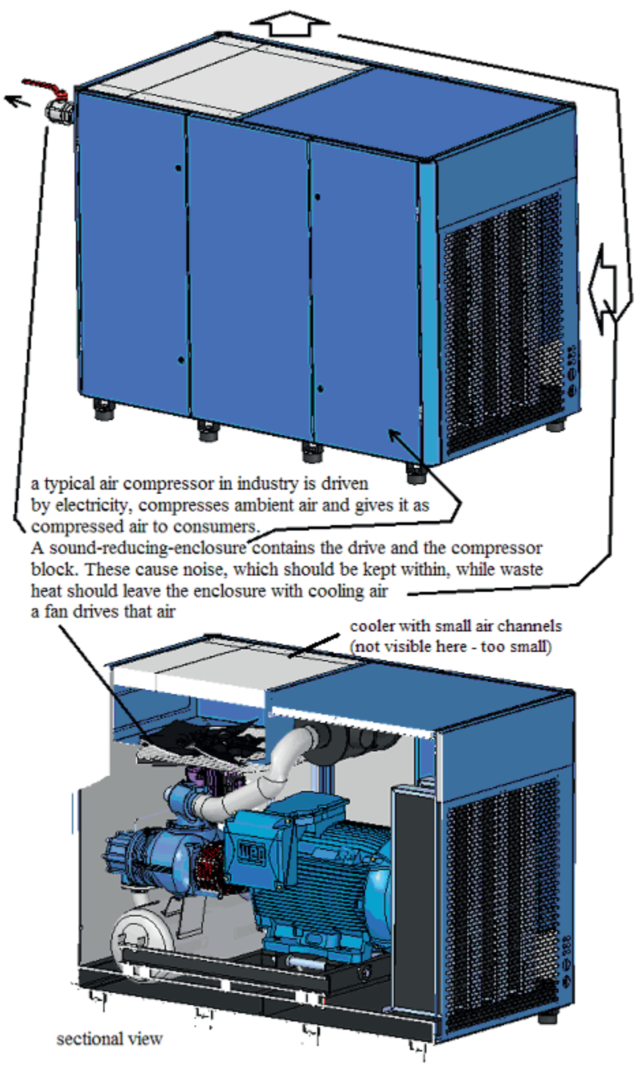
The application is clear: the flow of air inside (as compressed air) and outside (as cooling air) of a compressor is decisive for its function. The present practice of doing simple calculations for dimensioning and a lot of experimental work to develop a good design could be improved by calculating the airflow by CFD -> to find a better start for the experimental work. But the traditional way of buying a CFD program and a fast computer, having one or two developers do a training with the CFD for several months and finally starting to use CFD for designing ? That is a high (financial) hurdle. Then we were informed about a EU-program „Cloudflow" to promote cloud computing across the EU.

The plan was:
A compressor needs a better sound reducing enclosure -> what is a widespread problem in machine design: the sound reducing enclosure has the task to keep the noise inside, but the cooling air flow has to get through that enclosure, so there have to be openings. What makes the enclosure good for keeping the noise inside - small openings and cooling air channels with bends - makes airflow difficult. And sometimes there is a paradox: a sound reducing enclosure for strong sound reduction is designed with small openings -> the cooling-air-flow needs a bigger fan -> the fan causes more noise -> finally the machine noise is not reduced

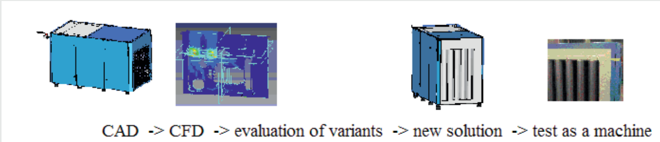
BOGE COMPRESSED AIR SYSTEMS

A Compendium about Cloud Computing in SME – the practical experience with “cloudy” CFD (computational flud dynamics -> calculating a flow instead of trial and error) in a medium-size enterprise

In some cases this optimisation task for air-cooled compressors takes a dozen experimental iterations - the compressor is in many configurations either too loud or too hot. A better understanding of the airflow clearly helps to find a good solution much faster. The CAD-model of the compressor to be improved is transferred into the CFD-program.

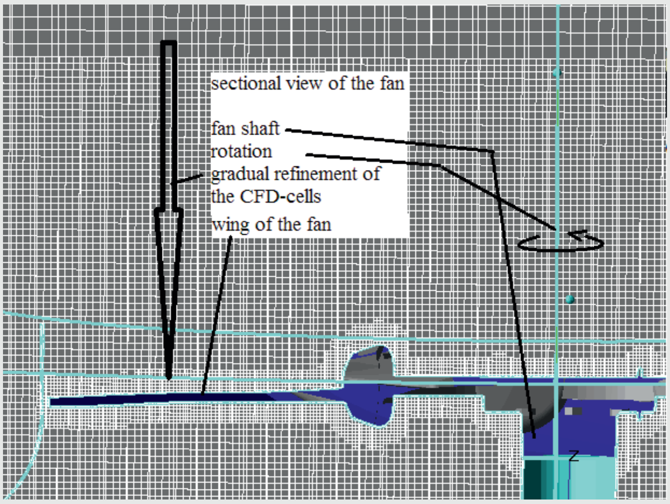


The CFD-program and a high-performance-computer (HPC) are cloud resources. The CFD-provider and the HPC-provider are available for questions through the internet, for example by video conference per internet. HPC makes computing faster and allows tackling even big problems. Several configurations of the design-optimisation-task can be simulated by CFD, so finally there is a design solution, ready for experimental test <- with few optimisation work to be done experimentally and -with the help of a better insight into airflow- it is a better solution than the traditional ones (just by experimental iteration / trial & error).

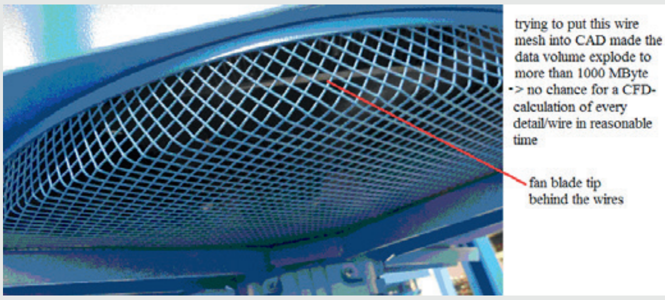


The work started. And then some problems appeared, that were not foreseen. The transfer from CAD to CFD was a bigger task. The CAD includes a lot of details, for example including many small screws in a 1000+ parts / 3x2x2 m³ compressor. When performing a 100% direct transfer, the CFD program (Flowvision by CAPVIDIA) created a very detailed model in the beginning, too big for a solution. It was necessary to strip the small and insignificant parts, and to close the screw holes manually (a 6-mm-hole in a sheet metal part gives a numerical stability problem).

A Compendium about Cloud Computing in SME – the practical experience with "cloudy" CFD (computational flud dynamics -> calculating a flow instead of trial and error) in a medium-size enterprise

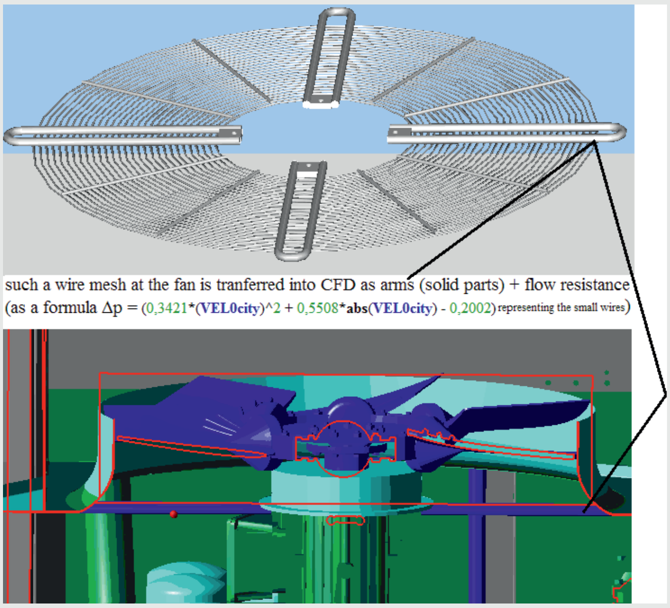


One important goal for optimization was fan power. In order to get the fan torque correctly, a fine mesh around the fan blades was necessary, increasing the numerical effort. Very helpful was the support by internet conferences with the CFD-supplier. Without that help, it had taken months of training for the us (as CFD users) to get the task done. But telling a problem to the CFD-supplier by an E-mail, having an internet conference next or next but one day, talking for some 1 or 2 hours, seeing the right way of operating the CFD (which parameters to enter, which variables to observe, where to click with the mouse) solves problems quickly. Recording these conferences for rehearsal helps. Some important details could not be included directly. A full, exact CAD-model including all wires of a finger-protecting mesh is too complex to include it into the CFD. So the flow resistance of the wire array is simulated in a separate model (with only the wire array and an airflow), and the result entered into the full-compressor-CFD-model as a flow resistance area with a formula of the kind $Dp = k1 * c^2 + k2 * c$ (the coefficients k1 and k2 being the results of the separate CFD model). Finally the full simulation was done. Although 24 processors were running in parallel on the high-performance computer, a full simulation took 10 days. This limits the number of possible enclosure variants to be simulated by CFD.



A conclusion from this is:
If it is not possible to eliminate small, insignificant parts automatically, it is often a better way not to do a direct transfer „full-CAD-model -> CFD" but to create a simplified CADmodel (with only the parts significant for airflow, small boreholes closed) and to transfer that into CFD. This will help for faster results. The transfer of grids, wire meshes, filters - important for the airflow, but with many small details - into a pressure loss coefficient should be done right from the beginning. But larger parts creating a wake influencing the fan, like holding arms, crossbars etc. have to be modelled as separate parts.

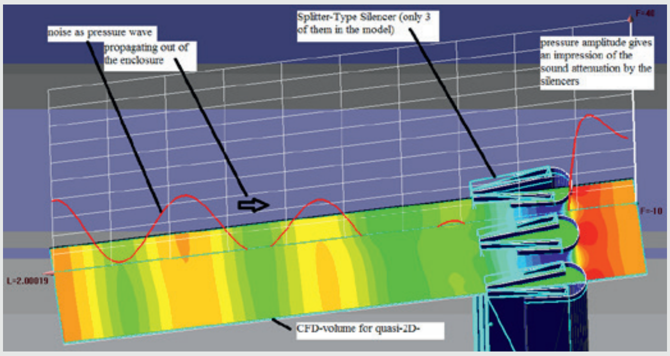
A Compendium about Cloud Computing in SME – the practical experience with "cloudy" CFD (computational flud dynamics -> calculating a flow instead of trial and error) in a medium-size enterprise



Internet conferences showed one problem: As usual for a production company, there is a company-LAN, protected with a strict firewall. Many programs for internet conferencing, known to have backdoors like SKYPE, are blocked. And the program used, „GOTOMEETING" has an update ca. every 2 weeks, everyone of them requiring a new configuration of the firewall. It would be better to find a stable internet conferencing program, that is safe (and includes a recording function).

There could be the idea, that sound is just pressure waves moving, and pressure waves can be calculated by CFD. Because a simulation of the sound propagation inside the sound reducing enclosure means going from steady (cooling air) flow to nonsteady flow and a mesh width of the computational grid of $< \lambda/10$ - what means < 33mm for 1000 Hz - this would result in increasing the computation by a factor > 1000. This was known before starting the project and therefore not intended. Sound reduction by the sound reducing enclosure should be done by the simple formula for an air duct with sound absorbing material (known as „Piening formula"), and this turned out to be a sensible rough calculation.

A sound calculation of the fan following a formula like that of VDI 3731 / Regenscheit, was intended, but did not match experimental results. The effect of wakes in the inflow to the fan is paramount. But a different problem in sound reducing enclosures turned out to be solvable. Openings in the enclosure allow noise to get out, and so should be small, but cause pressure loss in the cooling air flow. Splitter-type silencers installed at the openings help in reducing the sound outflow. Even better are streamlined splitter-type silencers, as they allow small openings with reduced pressure loss. Calculating their effect at high frequencies is not a problem, but at low frequencies a simple absorbing surface is not effective. Silencers at low frequency work with resonating chambers, and there is no formula for their silencing effect. But if it is possible to reduce that problem to a small channel with very few silencers inside, it can be solved as a 2D-non-steady flow by CFD.



It is necessary to test the ability of the CFD to simulate pressure waves -many CFDmodels include an artificial computational wave attenuation to improve convergence- but regarding that, the sound attenuation of splitter-type silencers with resonating chambers at low frequencies ($\lambda > 0{,}2$ of silencer length) can be calculated and optimised.

Measurements for the optimised compressor were made and showed a decrease of fan power (about a third), plus a decrease of fan noise (> 5 dB(A)), and a smaller decrease of the noise emission of the whole compressor. Having been shure about the better fan efficiency and having known a silencer's weakness at 500 Hz, it would have been possible to have a larger decrease for the whole compressor.

BOGE COMPRESSED AIR SYSTEMS

BOGE Druckluftsysteme GmbH & Co. KG
P.O. Box 10 07 13 · 33507 Bielefeld
Otto-Boge-Straße 1–7 · 33739 Bielefeld
Fon +49 5206 601-0 · Fax +49 5206 601-200
info@boge.de · **www.boge.de**

As far as now it is possible to say:
A cloud-computing approach to elaborate high-performance-computing-based methods for construction design is useful for small and medium-sized enterprises. It helps in overcoming the financial hurdle to start with that. Perhaps as an occasional instrument for complex construction tasks, perhaps as a way to get acquainted with that and finally buying the (in this case CFD-) program. Safety concerns are to be regarded carefully.

In this case there has not been a test by external „testing hackers" or an extensive evaluation by computer safety experts -> the costs for that would have exceeded the budget for this project. The safety issue was tackled in this way: The compressor model to be optimised is already on the market - so it is already known to competitors. The project was funded by the European community, the results are to be published, so a hackers gain would be minimal.

As an overview:
- Cloud computing is a good way to start with elaborate high-performance-computingbased methods for construction design, for small and medium-sized enterprises.
- It is essential to find a provider, who can help with operating the program at short notice.
- This help should be available by an internet conference, with your screen visible to the helping person.
- The internet conference program should be chosen with care - no program with backdoors like Skype, Lync ..., few updates to avoid firewall updating needs; a recording function to make an instruction video -from the explanation how to solve a problem- is very useful.
- Working on a remote computer is not a problem, as long as the data volume to be transferred is not too big.
- Safety should be taken seriously.

And especially for CFD:
- Do not overestimate the ability of high-performance computing, don't just plan to transfer a full CAD-model into CFD; it is more useful to have a model with only parts significant for the flow, detailing only for the significant parts.
- Wire mesh, fine grids etc. probably have to be modelled as flow resistance.
- For fan noise the influence of for example fan motor arms or other obstacles in the airflow to the fan is paramount.
- For the design of sound-reducing-enclosures around machines and devices needing cooling air - a widespread task in industry - CFD is useful and helps to find better solutions and reduce experimental work.
- In this case try to find better forms for splitter-type silencers, openings etc. in small separate models.
- A CFD-modelling of low frequency noise -which is difficult to calculate with simple wall sound absorption formulas- is possible, when there are some helpful circumstances - small, simple model, preferably 2D, $\lambda >$ 0,1 of machine / part length.