

Theoretical and Experimental Study for the Development of Efficient and Economic Stirling Regenerators

1 INTRODUCTION

Stirling engines make use of regenerative heat exchangers (regenerators) which are arranged in the gas transportation channels between the hot and cold parts of the engine system. The function of the regenerator is alternatively to store heat from the hot blow, and return it to the reversed cold blow of the oscillating gas flow.

The efficient heat transfer between the metallic matrix material of the regenerator and the working gas is of crucial importance for achieving high systems performances. During each engine cycle, a multiple of the net heat in- or output of the system has to be transferred alternatively between the gas and the matrix. This process preferably takes place within a regular, finely divided porous structure, providing a large contact area between the regenerator matrix and the fluid. Furthermore, the pressure drop, the dead gas volume as well as the heat conduction losses between the hot and cold side of the matrix must be kept as low as possible. These constraints readily explain the difficulty of optimising such structures, which need to be conceived in considering jointly the thermo- and fluid-dynamic aspects of these structures, as well as many further system parameters imposed by the Stirling system.

A particular challenge is then set by the economic manufacture of precise, highly repetitive miniaturised structures having a prescribed geometry.

Furthermore, the quality of the regenerator can hardly be assessed by direct measurements on the component itself, but needs to be deduced indirectly from physical parameters and from overall performance measurements made during the operation of complete Stirling systems.

1.1 State of the Art

Most of the presently existing, performant Stirling engines or cryo-coolers are made of stacked wire meshes (typically comprising 300 to 500 metallic sieves with wire mesh of 50 to 65 per cm (130 to 170 per inch) and wire diameters of approximately 0.050 mm). As an alternative, continuous, randomly arranged metallic filaments are used, which are compressed into the regenerator casing in a way to achieve a relatively homogeneous filling of the volume. Foil regenerators, consisting of a spirally wound stainless steel foil with random dimples to maintain a regular annular spacing between individual windings have also been tested.

From a thermo-fluid-dynamic point of view, neither one of these structures can be considered as an optimal solution for performant regenerators:

- x in the sieve structure, the gas flow wake behind each individual wire causes important pressure drop losses;
- x the compressed continuous wire regenerators are not perfectly uniform, neither locally nor across the entire assembly; the gas flow is not uniformly distributed and the required periodic heat transfer and storage cannot be achieved;
- x the laminated sheet structures ask for a considerable length for achieving the desired heat transfer. The heat conduction along these continuous sheets, as well as possible structural irregularities cause additional losses which may severely limit the performance of the regenerator.

Sieve regenerators are expensive to manufacture. As most engines presently are built in rather small series, the manufacturing cost for this component represent a critical cost factor, impeding the competitiveness of such Stirling systems.

In recent years, considerable effort was devoted in the U.S. to develop etched foil regenerators, essentially for cryogenic applications. These regenerators are fabricated from a single piece of foil material, into which slots and channels are photo-etched in precise patterns. The etched foil is then rolled upon itself to form a well defined porous structure with desired, clean aerodynamic flow passages. However, the considerable analytic and experimental efforts invested into these developments did not result in the expected improved regenerators.

1.2 Scientific and Technological Approach

In his thesis, Rühlich 1999 [1] analysed the flow of the working gas through various regenerator structures. He concludes that the array of wires should preferentially be provided with lenticular cross sections and arranged in a way to obtain flow channels with gradual changes in cross-section. A dimensionless characteristic number $Ru = NPH/NTU$ is determined, as the ratio of the pressure drop coefficient (NPH) and heat transfer coefficient (NTU); preferred profiles are those where this ratio Ru is minimum.

The numerical analysis of the flow through a repetitive mesh structure by means of existing CFD (Computational Fluid Dynamics) methods appears promising, particularly in view of determining repetitive profiled wire structures having low Ru -numbers.

The present work is a continuation of the developments undertaken by Rühlich. Improved computational methods have been applied for the analysis of standard sieve structures (serving as a reference), as well as of newly proposed regenerator structures.

Additional regenerator losses occur, which are caused by irregularities of the regenerator structure; of prime importance are wall effects or non-uniform in-and outflow from the regenerator. These losses need to be analysed separately from the structure itself.

Promising new regenerator structures were manufactured, in order to assess the practical feasibility of the envisaged solutions. The cost of manufacturing and the suitability of the selected processes for their series production can thus be evaluated.

A selected number of these regenerators were then installed in existing engines and cryo-generators. Their operation and experimental testing permit to evaluate the regenerator performance. The experimental results obtained are analysed by a variety of Stirling analysis programs.

A particular Stirling set-up operating at constant pressure was conceived. By its operation at constant pressure, no power is produced in this set-up, so that thermal losses are fully attributable to the regenerator losses (heat conduction and heat exchange losses, without being affected or masked by the heat throughput taking place in power engines. Thermal quality of the regenerator can thus be assessed with greater accuracy.

1.3 Objectives fixed for the Study

The overall goal of these efforts consist in providing decisive technical and economic advantages to the engines operating with the newly conceived regenerators against state of the art solutions. The manufacturing technology needs to be matured in considering future series fabrication of this component. The final product must reach an economic breakeven limit, permitting its use in e.g. decentralised CHP-units operating at competitive cost. The individual goals to be reached can be summarised as follows:

- x Improved understanding of the detailed heat transfer and pressure drop mechanisms taking place in a variety of regenerator structures;
- x improved understanding of auxiliary losses occurring in these regenerators;
- x optimised layout of the newly defined regenerator structures with regard to different systems applications;
- x improved regenerator performances as compared to state of the art solutions;
- x new regenerator concepts suited for mass production, resulting in reduced manufacturing costs for components showing at least equivalent performances as state of the art solutions;
- x Testing of newly conceived regenerators in
 - Stirling engines conceived as residential co-generation units;
 - Biomass-fuelled engines;
 - Stirling cryo-generators.

Depending upon the applications considered, prime importance is to be laid upon the improvement of the performance, or upon the economic manufacture of the regenerator.

1.4 Structure of the Report

The following subchapters describe 3 technical aspects which emphasise the above explanations on the capital role of the regenerator in Stirling systems. A summary description is given on new regenerator concepts which have been analysed and tested in the course of the project. At the same time, these subchapters introduce the technical part of the report, in which each partner describes the work of his responsibility:

Chapter	Subject	Partner
2.	CFD analysis of the flow through regenerator structures	TUD
3.	Regenerator dimensioning and auxiliary loss assessment	JPB + GC
4.	Regenerator Manufacturing Techniques	BEKAERT
5.	Stirling Engine Testing (2.5 kW engine)	MAYER & Cie
6.	Stirling Engine Testing (9 kW engine + regenerator test)	ACE
7.	Cryo-generator Testing	AIM
8.	Conclusions	all

1.5 Sankey Diagram of a Stirling Engine

The Sankey diagram illustrates the relative proportions of the heat flows taking place within a Stirling engine. For a typical layout the average heat flow Q_R periodically exchanged within the regenerator structure is about 4 to 5 times larger than the net heat input Q_{IN} of the engine.

The considerable heat exchange taking place within the regenerator explains the importance of fixing suitable layout configurations for this component. It plays a capital role for achieving good engine performances. A thermal inefficiency of only 1% occurring within this structure (based on Q_R) leads to a loss in relative engine efficiency of 4 to 5%. As will be shown later-on in this report, the additional pressure drop losses occurring in this component lead to supplementary losses of the same order of magnitude.

(Detailed calculations made by means of Stirling cycle programs. Results of calculations with such codes are presented in this report; however, the detailed description of the different models used in this study lies beyond the scope of the present report).

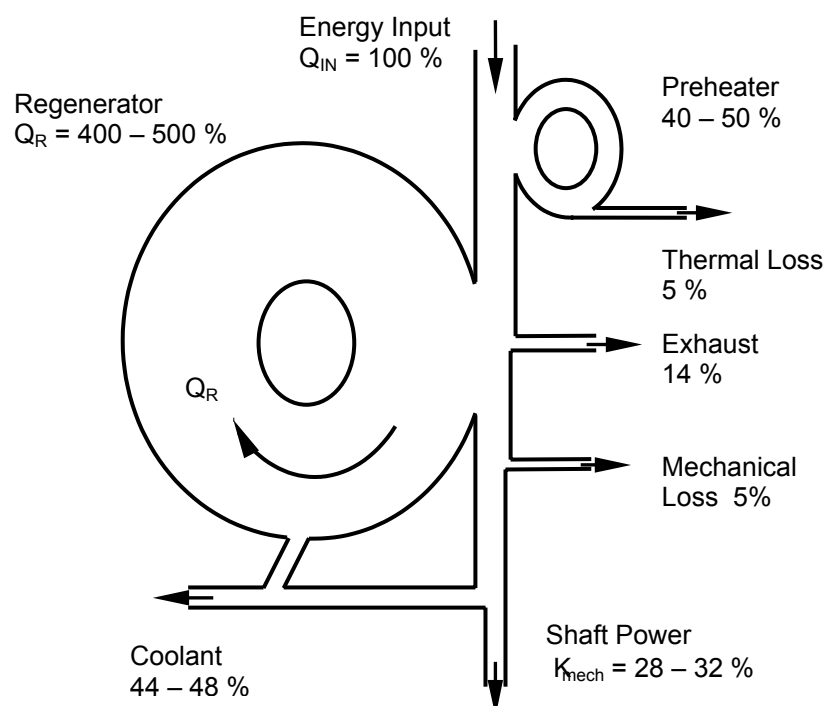


Figure 1.1: Sankey diagram of a typical Stirling engine

1.6 Comparison of various heat transfer surfaces

Heat transfer and pressure drop losses in the heat exchange equipment, particularly the regenerator can be expressed in form of dimensionless numbers. For an assumed steady flow, the quality of different heat transfer surfaces may suitably be characterised by means of a ratio of these dimensionless numbers (see also [I.Rühlich, 1999]):

$$NPH = \frac{\rho_f}{2} \frac{U_m^2}{D_h} f_f \frac{L}{D_h}$$

$$NTU = \frac{D S}{c_p m} \frac{4 k_i L}{D_h} \sim \frac{Nu}{RePr} \frac{4 L}{D_h} \sim St$$

$$Ru = \frac{NPH}{NTU} = \frac{f_f \frac{L}{D_h}}{\frac{4 L}{D_h} \frac{Nu}{RePr}} = \frac{f_f Re Pr}{4 Nu} = \frac{f_f}{4 St}$$

Favourable regenerator structures are characterised by low Ru - numbers. In figure 1.2, characteristic numbers of some well known regenerator structures are plotted against Reynolds number, based on hydraulic diameter and mean superficial flow velocity of the flow passing through them.

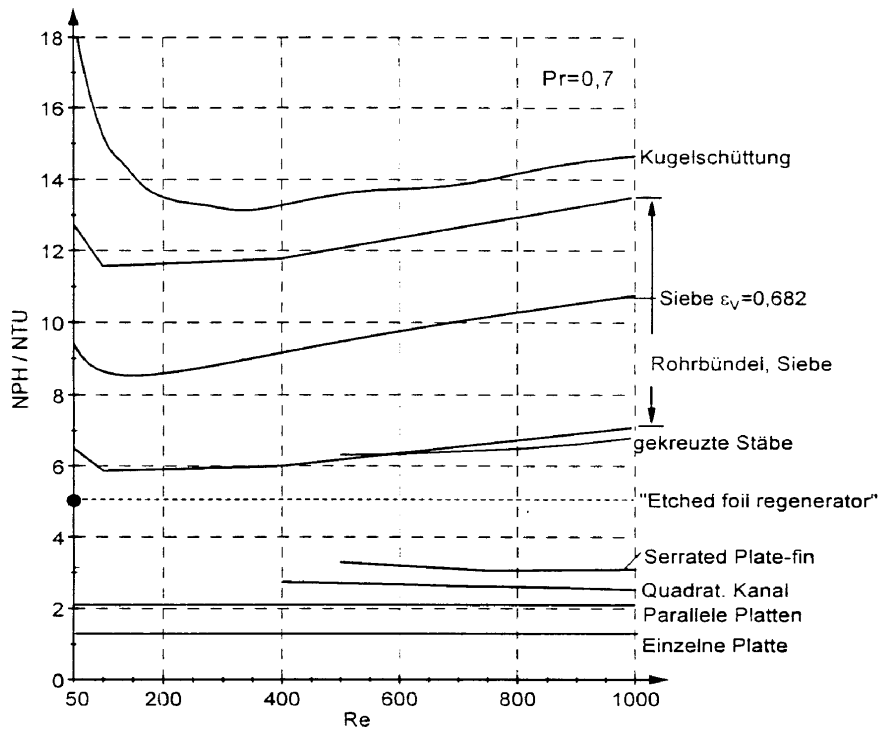


Figure 1.2: Performance factors $Ru = NPH/NTU$ for various regenerator structures (source: thesis of Rühlich, 1999 [1])

1.7 New Regenerator Concepts

In this project, 2 new regenerator concepts were developed and compared to existing and known regenerator structures.

A first solution aims at reducing the manufacturing cost to the lowest possible level. These regenerators consist of uniform, short metal fibres, which are sintered in a mould to provide the desired annular shape of given dimensions. The manufacturing process belongs to Bekaert, who was responsible for the development of the complete manufacturing process in view of obtaining a uniform product having the desired properties. Sample regenerators were produced according to predetermined specifications. These regenerators were used in Stirling engines and cryo-generators for determining the performance characteristics of these products.

The second solution aims at improving the thermo-fluid-dynamic characteristics of the regenerators. It consists of a wound annular cylinder formed as a sandwich structure by wrapping up a continuous, laminated sheet and a superposed sieve. The sieve acts as a rigid spacer to the wound sheet. The working fluid flows through the interstices left by the sieve between the continuous sheet layers. The surface of the sieve as well as of the continuous sheet participate in the periodic heat transfer and serve as heat absorbing materials.

In a first configuration, the sieve wires are oriented in flow direction and perpendicular to it. At the position of the perpendicular wires, the remaining restricted flow area represents approximately half the free area between the continuous sheets. These flow restrictions are of a similar magnitude as in the existing superposed sieve structures. For similar hydraulic diameters, the number of flow restrictions is reduced; it may thus be expected that the pressure drop may be lowered against those of conventional regenerator structures having the same heat transfer surfaces.

When arranging the sieves with the wires oriented at swept angles of $\pm 45^\circ$ relative to the principal flow direction, the total flow cross-section remains almost constant along the flow path. Pressure drop then results essentially from the meandering of the flow around the obliquely oriented, individual wires.

For these composed structures, no pressure drop or heat transfer characteristics are known to exist. A major study task therefore consists in the numerical analysis of the flow taking place through these structures and in assessing the required performance values. The desired optimum dimensions of such structures have to be determined. The corresponding regenerators need then to be manufactured and tested on Stirling systems.

These composed structures are assembled from existing, semi-finished products. As no sintering is needed, it is expected that such solutions can suitably be realised in an industrial process; series production should then result in affordable regenerators, at considerably lower cost than conventional regenerators with superposed sieves.

2. CFD ANALYSIS OF THE FLOW THROUGH REGENERATOR STRUCTURES

By Michael Hieke, Technische Universität Dresden

Objectives

The main objectives of this part of the presented study were to:

- x update the existing CFD calculation model and rationalize the in- and output of data by programming batch scripts
- x perform sample runs with simple structures and compare the results with known values
- x extension of the simulations to complex 3-dimensional structures
- x run steady state simulations
- x extend the simulations to unsteady state behavior
- x determine the improvements which can be expected from the optimization of the internal geometry

Content of the report

This chapter presents the results of the CFD simulations carried out within the study. It starts with a general introduction to CFD and its mathematical background. Then some samples are explained which were done in the beginning of this study. These results are shown and compared to known analytical solutions. They show the applicability of the software for the purpose of the study. In the next step, more complex three dimensional models are created, representing different internal regenerator geometries. These are based on sieves from the well-known stacked screen regenerator. A new type of a regenerator using wound sieves is introduced. Some basic assumptions for the simulations are explained in detail; in particular, the analysis of a small regenerator volume on which appropriate boundary conditions are applied. The difficulties of defining unsteady state boundary conditions are discussed. The post-processing algorithms are explained and some typical results obtained are presented graphically as well as numerically. Finally, the results will be discussed and summarized.

2.1 Introduction

“Computational Fluid Dynamics” (CFD) is a tool to compute the flow behavior, the heat transfer and other physical processes within a given domain. It is based on the solution of the flow equations in this domain with defined boundary conditions at the domain borders.

Computing programs for the numerical solution of flow problems exist for many years. At the beginning simple algorithms were used to solve specific problems. They required a thorough understanding in fluid dynamics. The applications of the early programs were restricted by the available computer hardware and were used exclusively in research. In the course of time, the algorithms were combined with software packages. Advanced solvers enable robust solution of the flow field in a reasonable time. In addition, the advances in computing hardware permit to simulate complex geometric contours for the flow models, with which a large variety of flow phenomena can be simulated. The calculation and the post-processing of the results can be carried out within an affordable computing time. CFD became an established industrial design tool, shortening and simplifying the design process for many new products. Today’s CFD-packages represent a convenient and accurate means by which the experimental testing and measuring effort can be reduced.

Mathematical Background

The simulation is based on solving the conservation equation of continuity, momentum and energy, known as Navier-Stokes-Equations. They are a set of differential equations developed in the early 19th century. There is no known general analytical solution, but the set of equations can be discretized and solved numerically.

Additional equations for heat transfer or other processes like combustion can be solved in conjunction with the Navier-Stokes-Equations. Approximating models are frequently used for describing these processes. An important example is the description of the turbulence by turbulence models.

There are a number of different solution methods which are used in CFD-codes. The most common is known as the Finite-Volume-Technique: the simulation domain is divided into small sub-regions, called

control volumes. The equations are discretized and solved numerically for each control volume. As a result, the distribution of each variable, like pressure or temperature at specific points throughout the domain can be shown. In this way, a full picture of the flow phenomena may be obtained.

Application in this study

For regenerators, flow simulation is the only way to reproduce and to understand the flow behavior. The internal regenerator structures are so small that the use of any measurement equipment would cause great disturbances to the flow. For this reason, measurements are possible at the in- and outlet of complete regenerators only.

One objective of this study concerns the investigation and the comparison of the flow within different regenerator geometries. Today's state of the art stacked screen regenerators are compared with other sieve arrangements. Three dimensional geometric models are created of the internal structure of each of the sieve arrangements of interest. On the defined unit cells, boundary conditions are applied and varied in order to obtain detailed information about the heat transfer characteristics.

The commercially available CFD-solver ANSYS-CFX will be used for all simulations. Up to now, the simulations are performed under steady state conditions. Some problems associated with unsteady state oscillating flow simulations are discussed in this report.

2.2 Pre-investigations and sample runs

2.2.1 General procedures

The general procedures for setting up, solving and analyzing a CFD-simulation are similar for all cases. The first step concerns the determination of the region of interest. It should be selected as small as possible and as simple as possible. The geometry of the flow volume can be treated by CAD. It consists of a simple three dimensional computer model of the simulation domain containing edges, inner surfaces and boundary surfaces. As a next step, a fine mesh has to be created out of the CAD-model. It represents the control volumes and reproduces the geometry in all details. Based on the generated surface meshes, a volume mesh is created which fills out the simulation domain with control volumes. There are different types of meshes: structured or unstructured. A structured mesh consists of hexahedrons, while unstructured meshes contain tetrahedrons and prisms. The main advantage of a structured grid consists in the smaller number of control volumes. On the other hand, it is very hard to completely mesh a complex geometry only with a structured grid. In such a case, the creation of an unstructured mesh is faster and easier. A combination of both mesh types is also possible: the core of the flow through is represented by a coarse, unstructured mesh, while the regions near the surfaces have a structured fine mesh. Such combined meshing grids are frequently applied for achieving a higher accuracy for simulation of the boundary layer.

Appropriate simulation models must be selected according to the flow problem to be investigated. Simulation models mean time scheme (steady / unsteady), the turbulence model (laminar / turbulent) as well as heat transfer models or other options like combustion. Additional options for the solver must be set; the convergence, the termination criteria and some monitoring values must be entered.

In the next step the boundary conditions for all boundary surfaces of the simulation domain are applied. For every surface the boundary type has to be defined as "Wall", "Inlet", "Outlet", "Symmetry", etc. Depending on the type, other settings have to be made: a "Wall" for example can be adiabatic, can deliver a heat flux or can have a temperature profile on it. Common definitions for an "Inlet" are velocity profiles, a given mass flow or an absolute pressure value. For an "Outlet" boundary similar definitions are possible. Simulations including heat transfer processes may comprise additional temperature profiles at the "Inlet".

After defining the boundary conditions, the simulation solver is started, running the simulation automatically. The iteration process can be monitored only by the indication of specifically selected values or by the residuals.

Once all termination criteria meet the set requirements, the simulation is finished and the results can be analyzed. The results can be displayed graphically by temperature contour plots, velocity contours or velocity vectors within defined cutting planes or surfaces. Otherwise, a numeric post-processing can

be programmed for obtaining values of the variables at specific points, averaged or integrated over an area.

2.2.2 Sample run - Heat transfer for a cylinder in cross-flow

For this sample run a geometrical model was created from scratch, the control volumes were meshed and the boundary conditions were defined. A solid cylinder is exposed to a gas flow perpendicular to its axis. The overall heat flux from cylinder to the fluid is defined, but local variations according to the flow behavior and local variations in the heat transfer coefficient may occur. Air is used for the fluid, while the solid is defined as copper. In Figure 2.2.1 the simulation domain is shown. On the left side the cold air is flowing into the domain with a constant velocity and a constant temperature profile. The upper and lower boundary surfaces are defined as adiabatic walls. The right side is the outlet for the fluid. The red circle represents the warm cylinder transferring heat to the fluid.

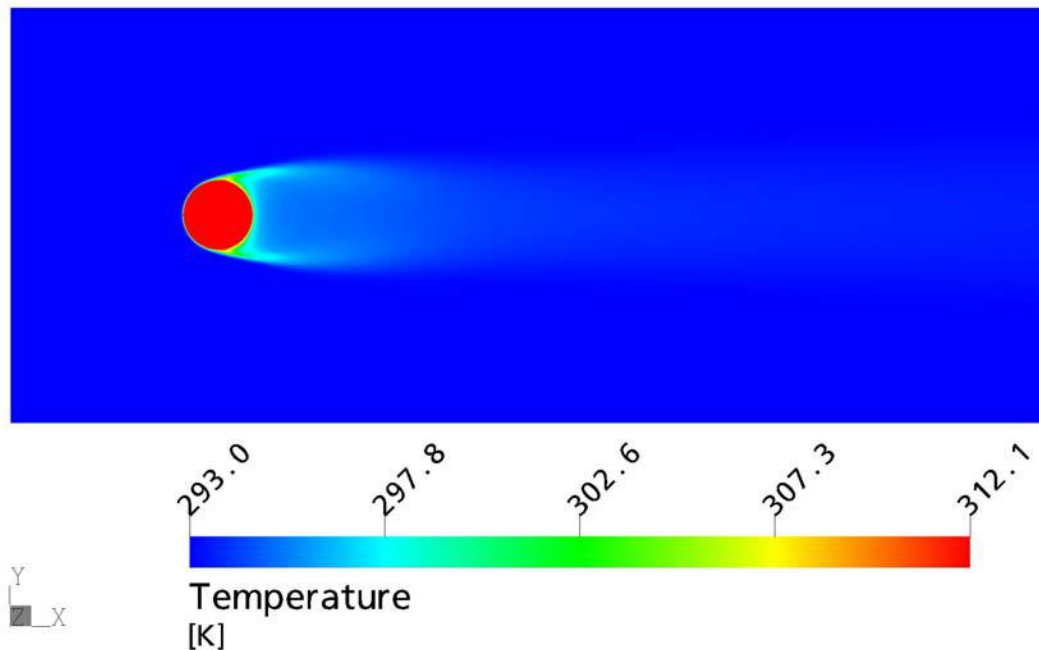


Figure 2.2.1: Simulation Model

The result obtained from the CFD-simulation is compared to an analytical solution from University Stuttgart [5].

With a given Reynolds number of

$$Re = \frac{c \cdot d}{Q}$$

an averaged Nusselt number can be calculated by

$$Nu = C_1 \cdot Re^{C_2} \cdot Pr^{C_3} \cdot \frac{Pr}{Pr_{wall}} \cdot \left(\frac{Pr}{Pr_{wall}} \right)^{0.25}$$

with the following coefficients

Re	C ₁	C ₂	C ₃
1 ... 40	0.76	0.4	0.37
40 ... 10 ³	0.52	0.5	0.37
10 ³ ... 2 · 10 ⁵	0.26	0.6	0.37
2 · 10 ⁵ ... 10 ⁷	0.023	0.8	0.4

The Nusselt number is used to calculate the average heat transfer coefficient

$$Nu = \frac{h D}{k_f}$$

Finally the CFD-Solution is compared to the analytical solution. With a Reynolds number of 3300 it can be shown that

for the analytic solution $h = 7.666 \text{ W / m}^2 \text{ K}$ and

for the CFD-solution $h = 7.633 \text{ W / m}^2 \text{ K}$

The relative error of the CFD-solution is calculated by

$$\text{Relative Error} = \frac{|h_{\text{CFD}} - h_{\text{analytic}}|}{h_{\text{analytic}}} = 0.43\%$$

The perpendicular flow to a circular cylinder is a very simplified model of the perpendicular flow to a wire within a regenerator sieve. A real regenerator consists of many, uniformly distributed wires. In spite of the more complex flow behavior the CFD-software should be able to handle the steady state heat transfer processes between solid and fluid accurately.

2.2.3 Sample run - Oscillating channel flow

Another test run was made for an oscillating channel flow. The example has to show the implementation and the correct solution of unsteady terms within the CFD-software. The geometry of the channel was meshed and the boundary conditions defined. The simulation was executed as unsteady state with time dependent variables. The given variables were the frequency f , channel height h , kinematic viscosity ν and a sinusoidal pressure gradient. Schlichting [4] shows an analytical solution of the Navier-Stokes-Equations for the oscillating channel flow:

An infinite long channel with x-axis in lengthwise direction and y-axis for the height inside the channel is defined. With a flow independent of the axial position x can be found that

$$\frac{\partial p}{\partial x} = \frac{\partial^2 u}{\partial y^2} - \omega u$$

With the assumption $u(y = \pm h/2) = 0$ at the walls and a harmonic sinusoidal pressure oscillation with a maximum K

$$\frac{\partial p}{\partial x} = K \sin(\omega t - \alpha z)$$

the solution for the time dependent velocity profile is given by

$$u(y, t) = \frac{K}{Z} e^{iZ\omega t} \frac{\cosh\left(\frac{y}{\delta} \sqrt{iZ\omega}\right)}{\cosh\left(\frac{h}{2\delta} \sqrt{iZ\omega}\right)}$$

The resulting analytical profiles are compared to the profiles from the CFD-simulation with identical boundary conditions. As an example, the results are shown for a channel with a height of $h = 0.01 \text{ m}$, a fixed pressure gradient and the physical properties of air for frequencies of $f = 1 \text{ Hz}$ and $f = 100 \text{ Hz}$. In Figure 2.2.2 the velocity profiles are shown over the channel height at a fixed time. The channel height can be read from the ordinate. The abscissa shows velocities and pressure gradients, normalized by their respective maximum.

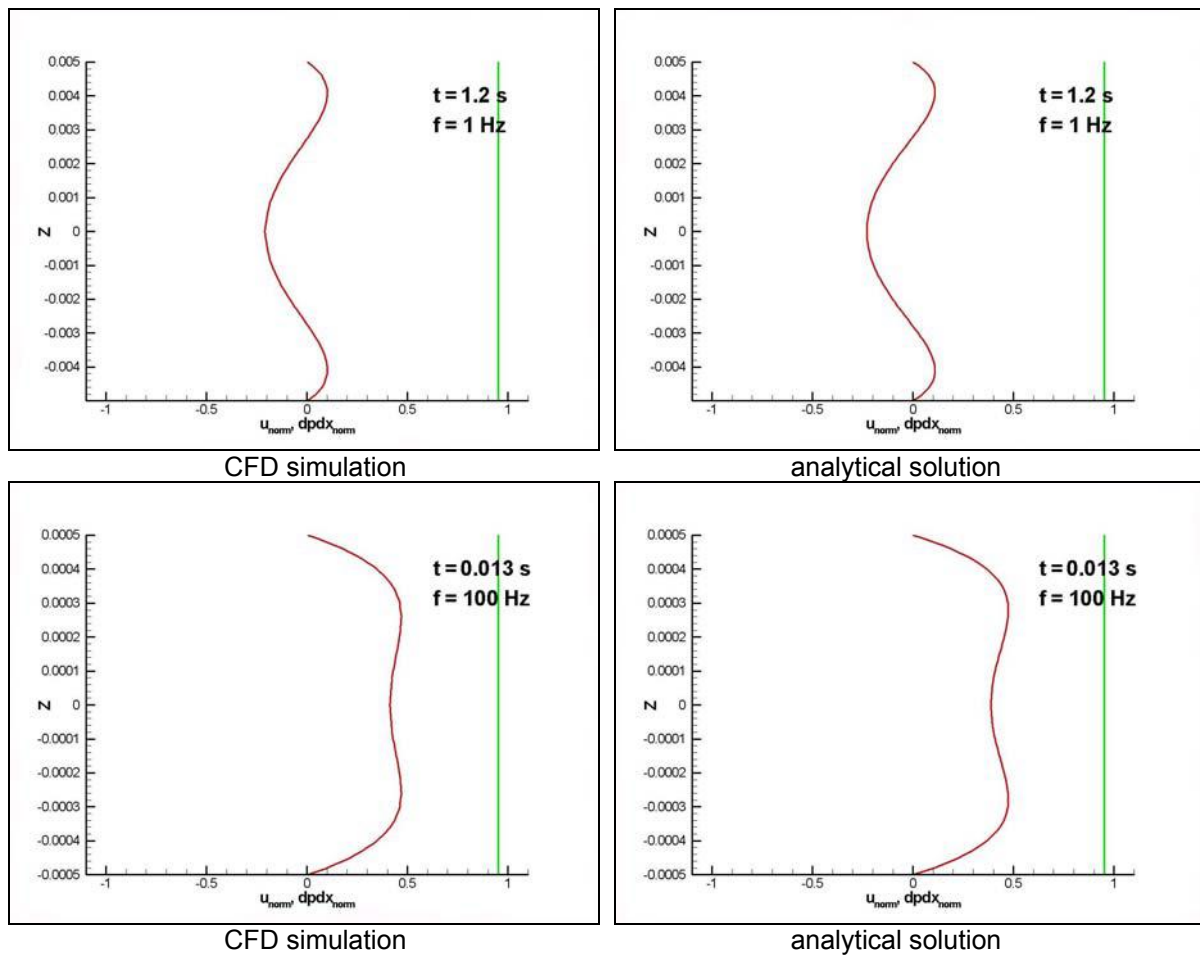


Figure 2.2.2: Normalized velocities and pressure gradients

CFD simulation yields the same results as the analytical solution under the given conditions. The unsteady inertia terms are correctly included in the simulation algorithms. Only with increasing frequency and decreasing channel height slight differences may be observed. Additional simulations would be required with variable time steps and finer local meshes. The performed tests revealed that appropriate starting conditions are essential for obtaining a fast and stable result. With bad starting conditions, the simulation needs more iterations for tuning into a stable solution.

Unsteady state flow simulations are feasible and should be applicable also for more complex geometries like regenerator sieves.

2.2.4 Sample run - Temperature swing in a cylinder exposed to oscillating cross-flow

In the next sample run, a cylinder is exposed to an oscillating cross-flow having different, oscillating temperatures. Unsteady heat transfer between the cylinder and the gas is taken into account, in addition to the unsteady flow simulation. According to Figure 2.2.2, a three dimensional model is created with a circular cylinder (diameter $d = 100 \mu\text{m}$) in a straight channel having two openings. The cylinder is defined by a "solid" domain with given heat capacity and heat conductivity only. Local solid temperatures can vary freely according to the stored heat and the transferred heat from the fluid.

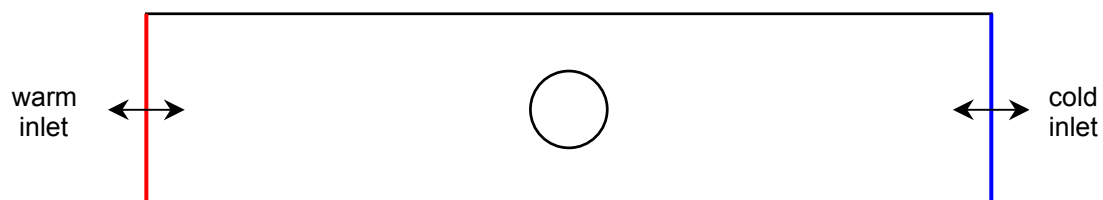


Figure 2.2.3: Model for unsteady heat transfer simulations

A sinusoidal oscillating flow is defined by an oscillating pressure gradient between the two openings. Warm gas flows from left to right, in the other direction the flow will be cold.

Figure 2.2.4 shows the average solid temperatures as a function of timestep for different operating frequencies. One oscillation period is divided into 50 timesteps for every frequency, so all curves can be plotted into one diagram. It can be shown that with increasing frequency the temperature swing in the solid becomes smaller. With shorter time for heat transfer, the temperature swing becomes lower.

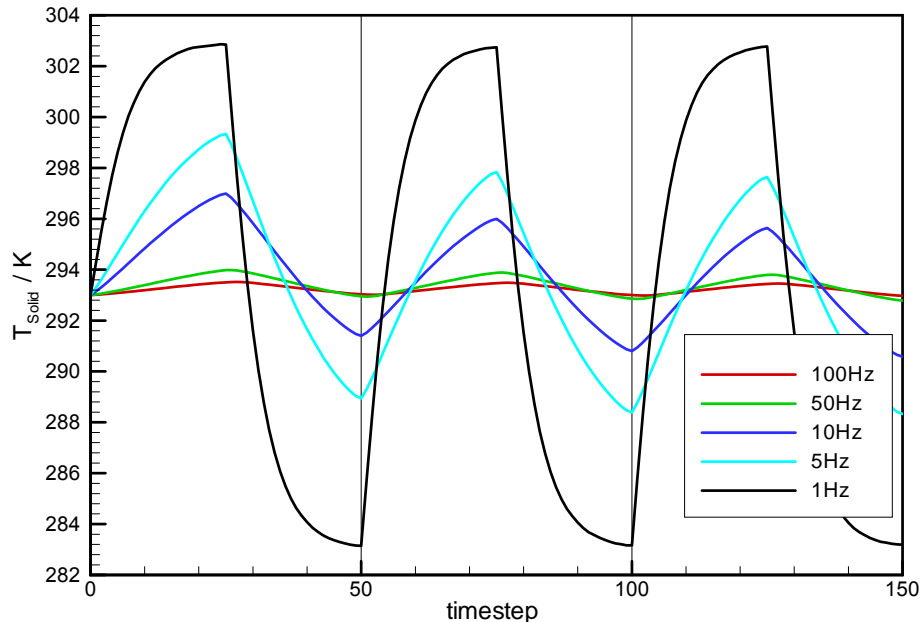


Figure 2.2.4: Temperature swing in a cylinder in cross-flow with different frequencies

The trend of the solid temperature distributions is as expected. The heat conduction and heat storage within solids is calculated for the oscillating flow with continuously varying temperatures. In unsteady state simulation, the temperature profile or heat flux at the wall surface does not have to be described. The CFD-software can handle oscillating heat transfer between solids and fluids and can calculate the dynamic heat storage within solids.

2.3 Steady state simulations in different regenerator structures

Several methods can be applied for the numerical analysis of the gas flowing through the regenerator matrix. One procedure consists in assuming the complete regenerator as a uniform, porous medium. Based on measured data, simplified expressions can be formulated for the pressure drop and heat transfer taking place within the porous medium. In- and outlet conditions are modeled in detail, in considering incident flow conditions. However, the detailed flow behavior within the matrix itself remains largely unknown.

Another method consists in analyzing only a small fraction of the regenerator matrix, within which the flow is simulated with all details, as well as the resulting heat transfer at the solid interfaces. Periodic flow conditions need to be assumed at the in- and outlet surfaces of the flow element under consideration. In this procedure, the flow within the regenerator volume is determined in detail; however, the exact behavior near the in- and outlet or close to the regenerator wall are not taken into account.

The present study is based on this second method of analysis: a small regenerator volume is considered in assuming periodic boundary conditions at its in- and outlet faces. The smaller the considered flow volume, the shorter will be the required computing time, respectively the narrower the nodes of the calculation network grid may be placed. With the available computer capacity, a trade-off between computing time and network density needs to be made, which for each flow geometry resulted in the use of approximately 500'000 nodes.

The smallest possible regenerator fraction to be considered can be determined rather easily. In the present study, all geometries of interest are based on the same type of sieve elements, as illustrated on the left-hand side of Figure 2.3.1. The smallest grid element having periodic faces along all 3 geometric axes is illustrated on the right-hand side of Figure 2.3.1. These sieves have a typical porosity of 75 %, i.e. that 75 % of the total cell volume is occupied by gas. The wire diameter is uniformly fixed at 100 μm .



Figure 2.3.1: Unit cell as smallest possible flow simulation volume

This unit cell may be exposed to the gas flowing in different directions. In the usual stapled sieve regenerators, the gas passes vertically through these sieves. Wake zones are formed at the downstream side of each wire. In addition, the repetitive change of the free flow area causes the gas to continuously accelerate and decelerate, thus giving rise to significant pressure drop losses.

As an alternative solution, it was considered to orient the flow parallel to the plane of the sieves. This may be realized by winding up sieve stripes into a cylindrical shape and by exposing it to a gas flow parallel to the cylinder axis. In such a geometry, part of the wires are oriented perpendicularly to the flow, entailing substantial flow losses. However, when cutting the sieve stripes diagonally to the wire directions, all wires of the cylindrically wound body are exposed at 45° with respect to the flow direction. In each flow plane, the wires have an elliptic section; reduced pressure drop losses can be expected as compared to perpendicularly exposed wires. In addition, the flow cross section remains virtually constant along the flow path and only insignificant local acceleration and deceleration of the flow will occur. Figure 2.3.2 illustrates the various flow directions relative to the unit cell.

Figure 2.3.2: Flow directions relative to the unit cell

These wound regenerators are realized with advantage by winding up the sieves concurrently with a plane foil stripe. In this way, the sieves do not intermesh, what considerably simplifies the winding operation. Each unit cell becomes exactly repetitive, what is not the case when the sieves intermesh one with each other layer. Also, the plane foils act as additional heat exchange surfaces, participating actively in the regenerator function.

Publishable Final technical report

Project Number: ENK5-CT-2001-00527

Title of Project: Theoretical and Experimental Study for the Development of efficient and economic Stirling Regenerators

Project Acronym: REGENERATE

Period of Project: 01.12.2001 – 30.11.2004

Date of Report: 28.01.2005

	1 st Contractor	2 nd Contractor
Name of the company:	Technische Universität Dresden Lehrstuhl für Kälte- und Kryotechnik	Bekaert Fibre Technologies
Contact person:	Anke Kahlenbach	Carl Vromant
Address: Country	Germany	Belgium
Post-code, Town	01062 Dresden	8550 Zwevegem
Street	Mommsenstraße 13	Bekaertstraat 2
Phone:	+49 351 46333574	+32 56 767096
Fax:	+49 351 46337247	+32 56 767100

	3 rd Contractor	4 th Contractor
Name of the company:	AEG Infrarotmodule GmbH	Mayer & Cie. GmbH & Co.
Contact person:	Ingo Rühlich	Wolfgang Vieweg
Address: Country	Germany	Germany
Post-code, Town	74072 Heilbronn	72438 Albstadt
Street	Theresienstraße 2	Postfach 20 15 80
Phone:	+49 7131 6212320	+32 7432 700377
Fax:	+49 7131 6212399	+32 7432 7009377

	5 th Contractor	6 th Contractor
Name of the company:	AC Energy ApS	Budliger Ingenieur - Conseil
Contact person:	Henrik Carlsen	Jean-Pierre Budliger
Address: Country	Danmark	Switzerland
Post-code, Town	2800 Lyngby	1228 Plan-Les-Ouates, Geneve
Street	DTU, Build. 403	18, Chemin des Aulx
Phone:	+45 45 254171	+41 22 7944889
Fax:	+45 45 930663	+41 22 7944889