Proceedings in
**MANUFACTURING
SYSTEMS**

Proceedings in Manufacturing Systems, Vol. 5 (2010), No.

SIMULATION POTENTALITIES FOR A RECIPROCATING COMPRESSOR

Christian LANDSCHÜTZER[1], Andreas WOLFSCHLUCKNER[2]

Abstract: *In the development of piston compressors for tire repair systems in the automotive sector it is increasingly important to use numerical simulation methods. On the one hand in order to reduce test costs, on the other hand to shorten the development time. The research objective of this work is to find the optimization potential of a piston compressor by the use of finite element- and MBS-programs. Several simulation models are created. The model for the thermal behavior provides the temperature distribution on the surface of the compressor. For the investigation of the vibration characteristics the natural frequencies from the compressor are identified. Furthermore the vibrations under operating conditions are analyzed. Result of the mechanical analysis is a dynamic visualization of the stresses in the crank-parts during a shaft rotation. These results make it possible to reduce the weight of the compressor parts. There are also made measurements and analytical calculations for validating the models.*

Key words: *piston compressor, tire repair system, simulation.*

1. INTRODUCTION

Small reciprocating compressors, as applied in the automotive industry for pneumatic suspension, vertically adjustable seats and tire repair devices are produced in large quantities and so the effort for business companies to find cost-detailed solutions is very high. Especially the avoidance of over sized designs is an important point. To meet this demand without a cost explosion in field tests, simulation tools should be introduced.

Using the example of a piston compressor for tire repair systems, shown which problems can be addressed through simulation, as well as the limits for an effective application of numerical methods. Therefore fields of interest the following problems and their solution statements are discussed:

- Analysis of the thermal compressor behavior (FEM)
- Natural frequency and operational vibration analysis (FEM)
- Mechanical analysis of compressor components (MKS/FEM)

To understand the operating mode of the compressor, some design features are listed: (fig. 1):

- The compressor is powered by a DC electric motor. (Item.1).
- The cooling of the cylinder housing (Item 5) and the motor is enhanced by a fan (Item 2).
- Cylinder housing (Item 5) and main parts of the cylinder head are made of a zinc alloy.
- A joint between the piston (item 8) and connecting rod (item 6) does not exist. In order to obtain a sealing of the swept volume in non-parallel position of cylinder and connection rod, the piston seal is manufactured from a flexible material.

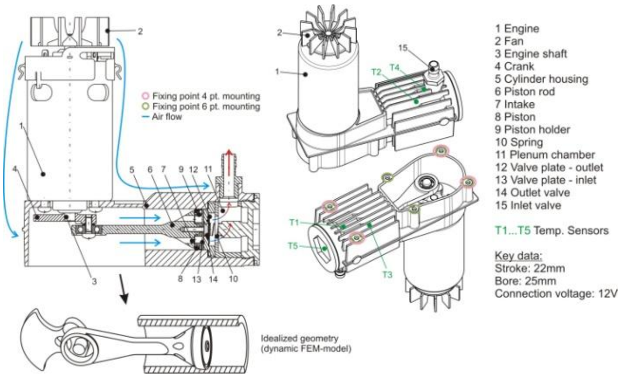


Fig. 1. Reciprocating compressor for tire repair systems.

2. THERMAL BEHAVIOUR OF COMPRESSOR

The surface area of the compressor is of particular interest. One important result is the temperature distribution on the surface which allows implementations of design improvements (e.g. location and number of cooling fins) and it must be possible to compare several compressor designs concerning the thermal efficiency by using the developed simulation model. That means the ability to emit heat to the environment should be optimized. Before developing the simulation model a analysis of the real system is made:

[1,2,]TU Graz, Institute of Logistics Engineering, Graz Austria
E-mail: *landschuetzer@tugraz.at*
Tel +43 (316) 873 7325
Fax +43 (316) 873 107325

2.1. Basic theory

The heat generated by internal heat sources of the compression machine, including also the electric motor, warms the components and is delivered partially through the outer surfaces to the ambient air. The remaining heat is dissipated with the compressed air. The following heat sources are identified in the compressor:

- electric motor (electrical losses and friction)
- compression heat
- frictional heat
- flow loss

For heat dissipation various mechanisms come into consideration:

- Convection (main part approx. 80%)
    - natural convection
    - forced convection
- Heat radiation (approx. 20%)
- Thermal conduction (is only important within the compressor)

The heat generated by the above-mentioned sources is dissipated through the surface of the cylinder housing (fig. 1, item 5). With the fan on the electric motor the convective heat transfer in a forced manner can be delivered to the ambient air. The fan on the electric motor supports the convective heat transfer to the ambient air.

2.2. Development of the thermal simulation model

The thermal model is created with the finite element program Ansys Workbench. Some advantages of Ansys Workbench respectively Ansys Classic are: [4]

- Interface to most popular CAD programs. In this case, the geometry data is created by Pro/Engineer without the need for a neutral data format to be accepted Ansys Workbench. Furthermore, maintaining the assembly structure and a possible update of the geometric data is possible.
- Solution of linear and nonlinear problems in structural mechanics, Fluid Mechanics, Acoustics, Thermodynamics, Piezoelectricity, Electromagnetism and combined tasks (multi-physics) is possible.
- Many element types for 1-, 2-, and 3-dimensionale finite element analysis.
- Ansys Workbench also has a modern GUI and improved algorithms for meshing and contact determination.

The following outline shows the basic approach to the model, especially simulation model development.

2.3. Data collection

For the data necessary to create the simulation model, there are different methods of survey. One group of parameters have to be identified explicitly in real compressor test runs:

- Total power consumption of the system (P_{el})
- output of the compressor ($\dot{m}$)
- ambient temperature (T_a)
- flow rate of the ambient air (w_a)
- temperature of the escaping air from the compressor (T_e)

And parameters found in engineering data reference books:

- Material data of the compressor
    - thermal conductivity l
    - specific heat capacity c_p
    - specific thermal expansion α
    - density ρ
- Characteristic parameters of thermal boundary conditions
    - heat transfer coefficient α_K (conv.)
    - emissivity ε (radiation)

2.4. Model building

Creation of the geometry

Except for some reasonable simplifications the original CAD geometry can be used. In this model no moving components are included (because of its low thermal mass their thermal impact is marginal).

Thermal loads

To determine the thermal loads on the inner surfaces, a power footprint for the system is useful. The system boundaries are shown in fig. 2. The difference between total power consumption and the thermal output dissipated by compressed air represents the heat loss $\dot{Q}_k$. $\dot{Q}_k$ (Includes heat of compression and friction heat) has to be dissipated by the surface of the cylinder housing.

- The electric power input is the product of the applied voltage U and current I:

$$P_{el} = U \cdot I \tag{1}$$

- The thermal output P_l dissipated via the hot compressed air is calculated by using the first law of thermodynamics:

$$P_l = \dot{m} \cdot c_{pl.} (T_e - T_a) \tag{2}$$

- And for the heat output $\dot{Q}_k$ follows:

$$\dot{Q}_k = P_{el} - P_l \tag{3}$$

c_{pl}...Specific heat capacity for air at constant pressure

$\dot{m}$ ...mass flow of air conveyed

The parameters U, I, $\dot{m}$, c_{pl} and T_a are almost constant during the compressing process. But the air temperatures T_e at the outlet branch vary strongly. So it is necessary to build time-discrete steps and to calculate constant values for $\dot{Q}_k$ respectively P_l at every time step.

Mentioned that a power footprint for the whole system is made, it is not known how to portion $\dot{Q}_k$ on the inner surfaces of the compressor to get realistic thermal loads. The thermal load allocation for first simulation runs is based on experience. If the validation of the simulation with these assumptions is not correct the parameters have to be adjusted. So this is an iterative process.

Fig. 2 shows the calculation of $\dot{Q}_k$ for several time steps and the areas with a constant surface heat flux.

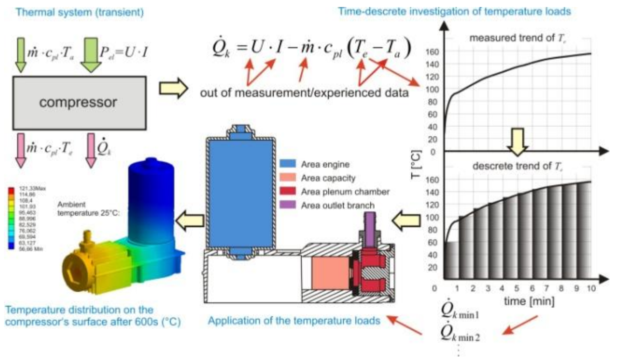


Fig. 2. Thermal behavior of the compressor.

2.5. Boundary conditions

Convective dissipation of heat

The principal part of the thermal output generated in the compressor is dissipated to the ambient air by convection at the outer compressor surfaces. Further parameter needed for defining the convective boundary conditions are the heat-transfer coefficient α_K, the ambient air flow speed w_a and the ambient temperature T_a. The heat-transfer coefficient α_K is calculated by an empirical equation which is valid for convective dissipation of heat (metal bodies flowed by air) [1].:

$$\alpha_K = 6,2 + 4,2 \cdot w_a \quad (4)$$

Heat emission by radiation

The proportion of the amount of heat emitted by radiation to the environment, in comparison to that which is dissipated by convection, is rather low because the heat transferred depends on the fourth power of temperature difference between cylinder surface and ambient air. This temperature difference is relatively low for radiation problems.

There are two determining factors necessary to define the radiation boundary condition completely. As indicated in the convection, the ambient air temperature is needed. And a second value is the emissivity ε which is a material constant.

2.6. Results

The simulation provides a graphical representation of the temperature distribution on the surface of the compressor. (Fig. 2), (power-on time: 10 min., delivering to a relative pressure of 2 bar constant). During this time, no stationary operating state is achieved because the thermal mass of the cylinder housing is too big for this.

The figure shows that especially the area around the cylinder head has the highest thermal stress. The surface temperature of the cylinder decreases in direction of the electric motor. These are important information for design improvements.

2.7. Validation

After the model definition is completed, the first simulation runs are feasible as a validation of the simulation results by comparing the levels found in the simulation with temperatures measured on the test bed. Fig. 1 shows the arrangement of temperature sensors and Table 1 the results after an on-time of 10 min. (delivering to a relative pressure of 2 bar constant).

Table 1

Comparison of measurement and simulation results

	Measured values [°C]	**Results from simulation [°C]**	**Discrepancy [%]**
T1	109,5	108,4	1,0
T2	103,0	105,5	2,4
T3	110,9	107,0	3,5
T4	109,2	108,8	0,4
T5	119,0	113,6	4,5

As can be seen clearly in table 1, the results are close together within 5 %. This variation is adequate, and thus the validation completed successfully.

2.8. Interpretation

The objective to analyze the thermal behavior of the compressor is completed very well. This allows precise statements for design improvements. In fig. 2 for e. g. the highly temperature-stressed areas are shown. These are also the areas where cooling fins are very effective. Model building (approx. 3 days) and computation effort (approx. 10 min. / Core 2 Duo 3GHz) are also in an economic reasonable time frame (this doesn´t include the effort to gain all necessary parameters). The validated simulation model can be used for several applications:

- Design optimizations (cooling fins, wall thickness etc.)
- Identifying influencing variables for load- and boundary conditions (for e.g. flow rate of the ambient air)

3. VIBRATION CHARACTERISTICS

Analysis of the vibration characteristics should provide insights into the deformational and kinesic behavior of the compressor. Primarily the cylinder housing is of interest. The orders of magnitude of the vibration caused deformation have to be investigated and if the deformations are relevant the direction for design improvements should be shown by the analyze result. To obtain the required simulation results two analyses are made:

- Modal analysis to identify the natural frequencies for determining operating condition (subcritical-critical-supercritical relating to the excitation frequency)
- Analysis of operational vibrations for calculating strains and deformation caused by mass force effected vibrations.

3.1. Model building

Because this simulation is also performed in Ansys Workbench, the basic elements of the thermal model can be reused. The basic model is identical with the thermal model (up to the meshing).

Loads

Load definitions are necessary only for the operational vibration behavior. For modal analyses loads are not needed. The bearing forces caused by the mass forces (for analytical calculation compare [2]) of the crank mechanism are applied for every coordinate axis. The angular phase shift f_P between the loads in x- and ydirection (90°) is important to consider. Fig. 3 shows the arrangement of the harmonic mass-acting forces of the crank mechanism. [5]

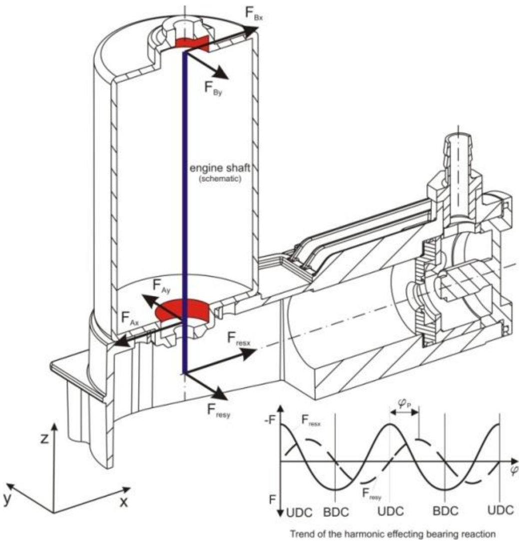


Fig. 3. Arrangement of the harmonic mass forces.

Boundary conditions

The modal and the operational vibration analysis are performed for two different mountings of the compressor. In Fig. 1 the position of the bearing points and the differences of the two mounting solutions are shown.

3.2. Results

Results of modal analysis are the lowest natural frequencies and associated vibration characteristics for each of the two compressor mounting types: (Table 2)

Table 2

Natural frequencies of the two compressor types

	natural frequency	**vibration characteristics**
4 pt. mounting	1333 Hz	Pivoting of the electric motor about the cylinder axis
6 pt. mounting	1858 Hz	Pivoting of the electric motor about the cylinder axis

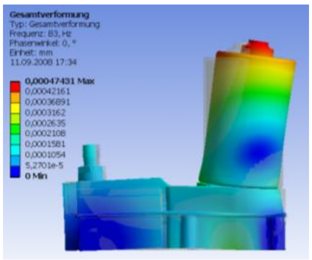
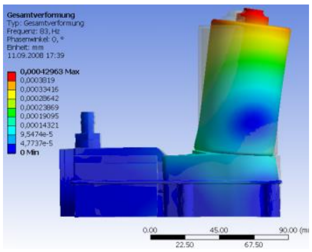
The electric motor is operated at 5000 rpm (equates to excitation frequency of 83 Hz). This means a sub-critical engine operating point and no risk of resonance effects primary.

The results of the analysis of operational vibrations are the component deformations and stresses due to harmonic mass forces. The total deformation for the two mounting types is very small and illustrated in table 3.

Using these results it can be verified whether the permissible deformations (or stresses) are exceeded or not.

Table 3

Total component deformation / operational vibrations

	Freq.	**Deformation behavior**
4 pt. mounting	83 Hz	
6 pt. mounting	83 Hz	

3.3. Validation

The validation of modal and operational vibration analysis is done by qualitative comparison of the real oscillation behavior with the simulation results. I.e. by simple observation it is noted shapes and frequencies at lowest resonances match. This type of validation, although does not provide full sureness wheter the simulation is correct.

3.4. Interpretation

Both objectives (Modal analysis and Analysis of the vibration characteristics) can be met by this modeling, with relatively little effort (similar to the thermal model). Since the natural frequency (1333 Hz) is much higher than the excitation frequency of the electric motor (83 Hz), there is no need to increase the accuracy of the simulation model.

4. MECHANICAL BEHAVIOR

Determining the stress in the compressor components is very important for revealing design weaknesses. The connecting rod with piston seated firmly on them to pay special attention, as these are very critical components in terms of function and service life. To represent the compressor realistically a MBS-software is used. This software has to be aware of flexible bodies and must allow the definition of contacts between the bodies (flexible-flexible/flexible-rigid). [6]

4.1. Model building

There are very few software tools that are suitable for the implementation of dynamic FE-simulations. For this purpose the program RecurDyn is chosen. Here are some key points about this software: [3]

- Recursive computation of multibody dynamics
- Core of the software is a technology called „Multi-Flexible-Body-Dynamics“ (MFBD)
  - This allows the combination of multibody dynamics (large displacements and rotations)

and non-linear finite element analysis based on a nodal approach, in contrast to the conventional modal reduction of the body.

- MFBD makes possible detailed simulation of contacts between flexible structures and rigid bodies and contacts between flexible structures of the model.

Creation of the geometry

Also for this model Pro / Engineer is used to generate the model geometry. The degree of simplification is very high. Thus, only moving parts and an extremely simplified form of the cylinder are contained in the model. In addition, the level of detail of the components of the crank mechanism will be reduced. This is necessary to keep the CPU time at a reasonable level. Fig. 1 shows a comparison of the original geometry and the simplified model.

Porting of geometric data

Since this model is constructed from two types of elements, that are flexible and rigid bodies, this must be also with the porting of the geometry data from Pro/Engineer are observed in RecurDyn.

Rigid bodies

To keep the computing time at acceptable levels, the bodies, in which stresses and deformations are not of interest, are rigid. These are the cylinder and the crankshaft. The port is done via neutral Parasolid files that are created in Pro/Engineer and imported to RecurDyn.

Flexible bodies

The port of flexible bodies is a little more difficult, as can be read only in RecurDyn meshed body geometry of the components. Flexible components are the connecting rod, piston and the piston holder. For porting the following steps are required:

1. Ports again produced from the parts in Pro/Engineer neutral geometry data and saved as IGES solids.
2. Now a FE mesh of the components is created. For this, the IGES data imported into the program Ansys (classic version). There the parts are meshed and the generated mesh is exported using CDB files.
3. At least the FE meshes are read into RecurDyn and flexible bodies are generated.

Configuration of the model:

The following joints and contacts need to be defined between the bodies:

1. Rotary joint between the environment and crankshaft.
2. Rotary joint between the crank pin and the large connecting rod eye. A connection between a rigid and a flexible body has to be created. This is done by use of a so-called FDR (Force Distributing Rigid) element, which divides the point force to surface element nodes of the connecting rod by rigid connections.
3. Contacts between connecting rod and piston and between piston and cylinder surface to ensure that the piston is centered in the cylinder and can run.

Loads and boundary conditions

The mechanism is set to a constant angular rate of the crankshaft and the piston with the gas force F_g, which is caused by the compression of air and applied by a function of the crank angle f. The gas force F_g has three different forms during one crankshaft revolution. Fig. 4 shows the final model for dynamic FEM analysis in RecurDyn with nodal flexible bodies.

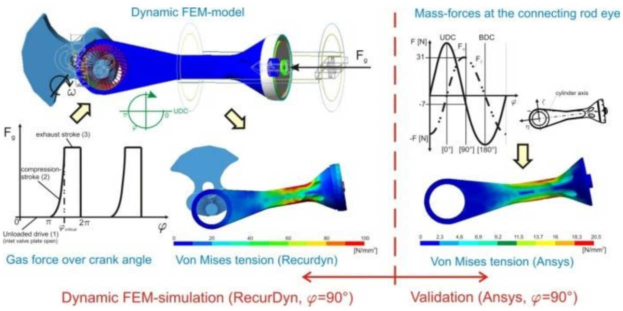


Fig. 4. Dynamic FEM analysis in RecurDyn and validation of the results in ANSYS.

4.2. Results

The dynamic FEM simulation provides the stresses and deformations during one revolution of the crankshaft. Of interest are the behavior of the connecting rod and crankshaft. RecurDyn allows an animated presentation of results. Fig. 4 shows the picture of such an animation for one position of the crankshaft (The cylinder is hidden). Particularly striking is the high amount of stress on the edges of connecting rod. This amounts to over 100 N/mm². This seems very high and therefore care is taken in the validation of the model especially at the height of the component stresses.

4.3. Validation

Validation of the RecurDyn model uses a quasi-static stress analysis carried out of the connecting rod. In a first step, the bearing forces, which act in the eye of the connecting rod, are determined by MBS-analysis with rigid bodies. The calculated values are passed to a static analysis for a crank angle of 90° (Fig. 4). This is achieved in any CAD program with CAE extension, such as Pro/Mechanica within Pro/Engineer. Here, the static FEM-analysis is performed in Ansys. For this analysis, the piston end of the connecting rod is fixed, respecting the mass inertia of the connecting rod simulation´s resistance to movement. Fig. 4 shows the results of this analysis. Comparing the stress distribution with that calculated for the same position of the crank angle by RecurDyn, the stress distribution equals, but the absolute value of stresses shows very large differences of about 1:5. The suspected reason for this difference is the complex contact situation (friction and geometric characteristics) between cylinder and piston which is very hard to represent in an adequate manner. A successful validation of this model is not possible with this approach.

4.4. Interpretation

The required objective (Mechanical analysis of compressor components) is failed, because a positive validation was not possible. Also the effort for model generation (about five days) and the computation time of the model (about two days) are very high. Additionally users must have good knowledge of software fundamentals and basic theory.

5. EVALUATION OF MODELS AND OUTLOOK

The results of thermal and vibration simulations provide valuable insights for further compressor designs and other improvements. In this domain the FEM software is already advanced and the use of such programs makes also an economic sense for smaller companies.

In the field of dynamic FEM simulation, several improvements are necessary to make available such analysis tools for a broad community of users. In particular, the improvement of the interfaces to other programs (CAD software) and a modern user interface are important points to these very capable programs make more widely accessible to users which works well.

Table 4 shows a qualitative assessment of the three different models with respect to several criteria.

Table 4

Qualitative assessment of the simulation models

	Modeling effort	Computing time	Software	Adjustment for geometry optimization processes	Model flexibility on new load and boundary conditions	Validation	Applicability of validated Simulation Model	Benefit/ Effort
Thermal FEM-Model	Low/ Interface to CAD, GUI	Low / 10min, Core2 Duo	Ansys WB. V11	Excellent Very good Interface to CAD	Excellent	Successful/ Temperature measurements	Thermal optimization of the compressor	Good
Model for the analysis of vibrations	Low/ Interface to CAD, GUI	Low/ 20min, Core2 Duo	Ansys WB. V11	Excellent Very good Interface to CAD	Excellent	Successful/ Qualitative comparison	Minimize vibration caused stress	Excellent
Structural dynamic FEM-Model	High/ Laborious contact def.	Very High/ 10h, Core2 Duo	RecurDyn V6.3 Ansys WB. V11 Pro/Mechanica	Difficult Rebuilding of the model is needed	Good	Not Successful/ Static FEM analysis	No re-use possible	Unsatisfactory

REFERENCES

[1] H. Recknagel, E. Sprenger, E. Schramek, *Taschenbuch für Heizung + Klimatechnik* (Handbook for heating and air-conditioning technology). Oldenbourg Verlag, 2000.

[2] R. van Basshuysen, / F. Schäfer, *Handbuch Verbrennungsmotor: Grundlagen, Komponenten, Systeme, Perspektiven* (Handbook for combustion engines), Vieweg+Teubner Verlag, 2005.

[3] www.functionbay.com / FunctionBay, Inc. / September 2008

[4] www.ansys.com / ANSYS, Inc. / August 2009

[5] H. Dubbel, *Dubbel. Taschenbuch für den Maschinenbau* (Handbook for mechanical engineering), Oldenbourg Verlag, 2000.

[6] W. Schiehlen, *Multibody Systems Handbook*, Berlin, Heidelberg: Springer, 1990.