Conference on Modelling Fluid Flow (CMFF'15)
The 16th International Conference on Fluid Flow Technologies
Budapest, Hungary, September 1-4, 2015

DYNAMIC MESHING STRATEGIES TO MODEL FLUID FLOW IN ROLLING PISTON COMPRESSORS

Balázs Farkas[1], Viktor Szente[2], Jenő Miklós Suda[3]

[1] Corresponding Author. Department of Fluid Mechanics, Budapest University of Technology and Economics. Bertalan Lajos u. 4 - 6, H-1111 Budapest, Hungary. Tel.: +36 1 463 2464, Fax: +36 1 463 3464, E-mail: farkas@ara.bme.hu
[2] Department of Fluid Mechanics, Budapest University of Technology and Economics. E-mail: szente@ara.bme.hu
[3] Department of Fluid Mechanics, Budapest University of Technology and Economics. E-mail: suda@ara.bme.hu

ABSTRACT

By the nature of their working process, the clearances within the volumetric compressors are subjected to complex deformations. Therefore, deforming mesh has to be used to simulate transient effects with CFD codes. Using dynamic meshing makes the simulation of essential phenomena such as leakage flows and heat transfer effects quite challenging because the desired mesh quality close to the boundaries and within the narrow seal cavities is not easily maintainable in every instant as a result of deforming numerical domain. To achieve the desired mesh quality several methods are available. ANSYS Fluent has inbuilt smoothing and re-meshing options which give relatively limited control to the user. To overcome this issue predefined meshes can be used which can assure appropriate resolution at certain positions of the rolling cylinder which can help to keep the mesh quality within acceptable limit for the whole cycle. For more elaborate solution the mesh can be also fully controlled by using user defined functions which allows for creating unique meshing algorithm for the given problem. The aim of this study is to compare the accessible models within ANSYS Fluent and find a proper meshing method which allows to provide accurate prediction about the performance of a rolling piston compressor in the early stage of the design process.

Keywords: CFD, dynamic meshing, rolling piston compressor

1. INTRODUCTION

Because of the urgent need to use environmentally friendly energy resources, many new progressive solutions have been developed or are under ongoing development in recent years. Part of these solutions extract electric power from geothermal sources which can be an efficient way when easily accessible high thermal energy sources are available. However recently further efforts have been made to extract electric power from low enthalpy heat sources. Same sources are already widely used for air conditioning and heating purposes mainly for small individual households. Nevertheless, power generation using low enthalpy heat sources is more challenging since the low temperature ratio already restricts the thermal efficiency of the applied thermodynamic cycle. Therefore, the efficiency of the individual components used to realize the thermodynamic cycle has to be as high as possible to provide a cost efficient solution, even if waste amount of heat is available from the source. In case of conventional solutions the compressor and the expander are key elements of the system. To get the highest efficiency, choice of the construction type for the given purpose is as important as the careful design and manufacturing is. Here the traditional rolling piston type compressor design was used as a platform and was further improved as it is shown in Figure 1 to achieve sufficiently high performance. The vane

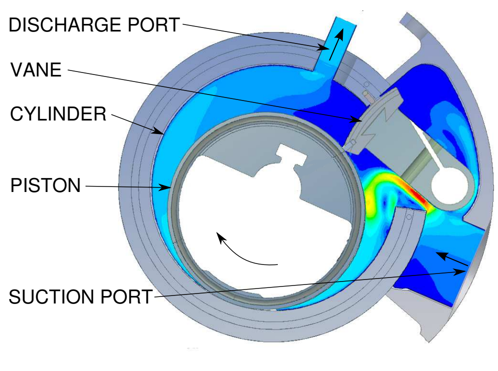


Figure 1. Velocity distribution within the rotating piston compressor designed by Magai [1]

which separates the high and low pressure chambers was redesigned and it is directly driven from the crankshaft of the piston which provides constant gap between the vane and piston, regardless the rotation speed without using extensive constrictive force and increasing the friction induced losses. The piston is solidly mounted on the crankshaft and does not roll along the inner surface of the cylinder. This design makes it similar to the trochoidal compressors, which do not require extremely small and expensive manufacturing tolerances because of the closed sealing border of the compression space and neither do they need oil for sealing [2].

To verify the theory, Computational Fluid Dynamics (CFD) tool was chosen to estimate the performance of this new design. Although CFD is known to be a very economical tool during evolutionary design process [3], most of the positive displacement compressor related studies are focused on developing and using concentrated parameter models since these components were considered as mainly thermodynamic devices. But within the past decade the havoc caused by fluid dynamics in destroying the efficiency has started gaining attention due to the pressing need for making these machines more energy efficient and reliable.

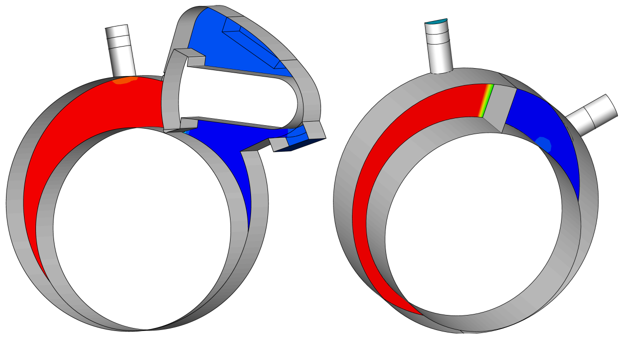
In 2004 two papers were presented at the International Compressor Engineering Conference in Purdue by the same company introducing a parametric CFD study related to the effect of the design of the notch [4] and suction piping [5] in meaning of noise and efficiency in case of a given double discharge compressor architecture. In both cases STAR-CD, a general CFD software was used for the simulations. The applied meshing techniques were not discussed in details but figures presented in [4] show a quadratic mesh within the cylinder with extended region of highly skewed elements around piston-cylinder gap and close to the vane. Despite these anomalies, the results resembled well the theory. In [5] where experimental tests were also conducted, the numerically predicted efficiency was also aligned well with the test results to verify the theory. More importantly, the trends in the change of performance parameters caused by geometry modifications, were predicted in good fashion.

In a study from 2010 by Liang et al. [6] two different kind of rolling piston compressors were investigated which were implemented with pressure activated discharge valves. The same solvers for the solution and governing the motion of the deforming mesh were used as in the above mentioned studies, but no further details were enclosed either. The predictions were also compared with experimental results. The error of the cooling capacity and power with the simulation and experiment test were less then 7%. The pressure rise vs. cranckshaft angle diagram of the simulation result is also claimed to be very close to the experiment test, although evident discrepancies can be observed on the presented graphs. Hui Ding et al. [7] from 2014 presented a new approach for simulating the applied discharge reed valve. For the simulations the PumpLinx CFD package were used which is specially targeted for simulating volumetric machines. According to the figures the applied meshing and re-meshing algorithms result in a good quality mesh within the whole computational domain where no highly distorted elements could been picked up. Here no further details about the applied meshing methods were enclosed either. Impressively, a whole revolution was claimed to be simulated within five hours on a general purpose quad-core Intel Xeon CPU at 2.67GHz machine. The predictions seemed to be aligned well with the theory, but obtained results were not compared to experimental test results.

Although deforming meshing seems to be ineludible, in the study of Brancher & Deschamps [8] the effect of the rotating rolling piston on the suction and discharge losses was estimated by steady 3D CFD solutions at different crankshaft angles. The predicted effective flow area and effective force area coefficients were implemented into a lumped parameter model. As a result significant improvement in the performance estimation were confirmed by experimental data.

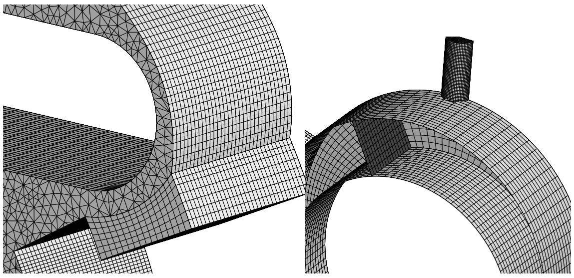
2. SIMULATION MODEL

3D representation of the of the numerical domain resembling the Magai design compressor is presented on Figure 2(a). Since in this case the model compressor is symmetrical, the numerical domain was divided into two and only one half of it was modeled and it was completed by using symmetry boundary condition at the symmetry plane. The initial mesh was created using ANSYS Workbench but during the simulation the mesh was controlled by Fluent's dynamic mesh models [9]. The domain was meshed by triangular elements on the frontal faces which were then extruded to the axial direction parallel to the crankshaft creating wedge type elements as it is shown on Figure 3(a). During the transient computation the mesh was only regenerated on the frontal base mesh and this frontal mesh was than projected to the parallel mesh surfaces similarly as the base mesh was initially created. In ANSYS fluent terminology it is referred as 2.5D approach. The node points on the cylinder remained station-



(a) (b)

Figure 2. Comparison between original Magai type compressor geometry (a) and the redesigned model compressor geometry (b)



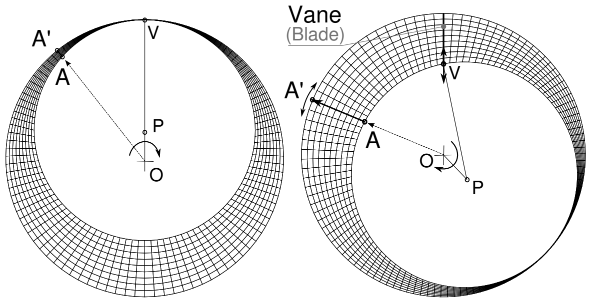
(a) (b)

Figure 3. Comparison of the numerical mesh for original Magai type compressor geometry (a) and for the redesigned model geometry (b)

ary and the nodes on the piston and vane surfaces followed rigid body like motion, which means that they kept their position respect to each other meanwhile the piston and the vane were rotated along their axes. To control the node points on the base surface spring/Laplace based smoothing and re-meshing algorithms were used. Both algorithms are part of the Fluent's dynamic mesh models. As the distortion of the cells increases by the large scale movements, ANSYS Fluent agglomerates cells that violate the initially defined skewness or size criteria and locally re-meshes the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells with the solution interpolated from the old cells. Otherwise, the new cells are discarded and the old cells are retained [9]. This solution results in relatively easy model setup since only the rigid body motion of the moving parts has to be predefined by the user by external User Defined Functions (UDF's) and the rest is taken care by the Fluent's inbuilt algorithms. The maximum size of the cells in the tangential direction on the surfaces were limited by the gap between the moving elements to gain appropriate mesh quality within the small clearances. The deforming grid has to provide a connected domain of constant area which means that no parts of the flow domain can be isolated from the rest, therefore even in the smallest gap a one cell high layer has to be remained. Rest of the base surface is discretized with constant size elements which resulted in unreasonable resolution outside the boundary regions. Several attempts were made to preserve the initial mesh quality but in every case after the first revolution of the piston the computational domain got overlayed with homogeneous size distribution elements. This method resulted in eminently time consuming simulations. The simulation for one revolution took almost five days on a quad-core i7 k2600 processor when the mesh counted $2 \cdot 10^5$ cells in average, and the simulation was parallelized for all four cores available in one processor. The idea to use predefined meshes at given crankshaft angles were dropped when attempts to model the check valve were implemented. This is because unlike in case of a piston which rotates at a constant speed the position of the check or reed valve at a given crankshaft angle cannot be predicted in advance [10].

2.1. Redesigned model and new re-meshing approach

Because of the unexpectedly long solution time, the old model was replaced and the new computational domain is shown in Figure 2 (b). Here the vane was replaced by an impermeable blade with infinitesimal thickness which significantly reduces the complexity of the original geometry. The new discretization is presented in Figure 3 (b) in comparison with the original mesh. The mesh contains only hexaghedral elements. The position of the mesh nodes within the cylinder were fully controlled by an external UDF using designated macros provided by ANSYS Fluent. The re-meshing process is illustrated in Figures 4 (a) and (b). During the rotation of the



(a) (b)

Figure 4. Illustration of the new meshing process by two meshes taken at (a) 0^o **and (b)** 135^o **crankshaft angles**

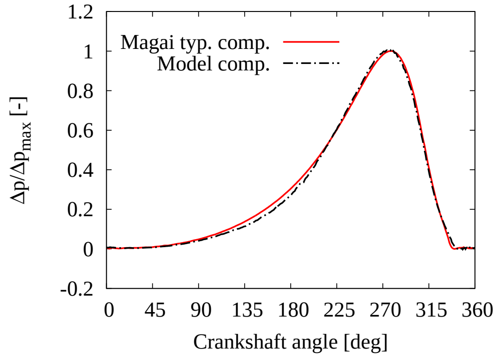
piston the axial position of the nodes do not change, which means that the nodes do not move parallel to the direction of the crankshaft. The motion of the nodes on the piston resembles a rigid body like motion, where the point *P*, which is the center of the piston, point *V*, which is the bottom point of the vane on the piston, and point *A*, which is an arbitrary node point on the piston surface, retain their position in respect to each other at each instant of the cylinder motion. This rigid body motion is described by the motion of the imaginary *O*-*V* rod which is part of the simple *O*-*P*-*V* crank mechanism where point *O* is the center of the cylinder and also defines the position of the axis of the crankshaft. The crankshaft angle is defined by the angular position of the *O*-*P* section which rotates around point *O*. The movement of point *V* is restricted to the vertical direction which also means that the blade keeps its vertical position during the rotation of the cylinder. This motion resembles the motion of a piston in a hinged vane compressor [11]. The position of an arbitrary *A'* node point on the cylinder surface is than defined as the projection of point *A* from the piston surface along the straight *O*-*A*-*A'* line. The rest of the points between *A* and *A'* are distributed equally within the straight *A-A'* section. This equal distribution is optional and not restricted by the method. Also, the method can be adapted to different blade thicknesses. Using the same settings discussed in the following section the simulation of one revolution took approximately twelve hours on the same architecture using just one core of the quad-core processor. The mesh holds constant number of $2 \cdot 10^4$ cells. At this point, parallelization was not implemented in the prewritten UDF code, since the achieved simulation time was already acceptable. Also, by running different cases in parallel, the available computational capacity was suitably utilized by the simultaneously running serial processes.

3. COMPARISON BETWEEN THE DIFFERENT MODELS

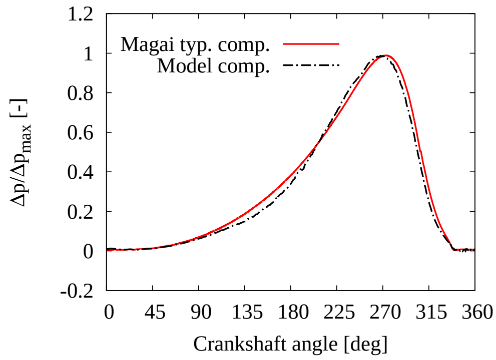
For performance estimations, the compressor was throttled back by adding a porous layer upstream from the exit of the outlet pipe. The porous media was modeled by the addition of the Darcy-Forchheimer type source term to the standard fluid flow equations. For both models, the outlet pipe was connected across a non-conformal mesh interface [9] to the cylinder and the geometry of the outlet pipe was identical in both cases. In case of the simplified model the inlet pipe was also connected in the same way to the cylinder. The inlet and outlet pipe tangential position for the simplified model was defined so that compression process would have the same extension as it is in the case of the original model. The solver was used with the standard settings. To model turbulent quantities, realizable k-ϵ turbulence model was used, both at the inlet and the outlet the ambient temperature and pressure was imposed as boundary conditions. For the preliminary computations the walls were considered to be adiabatic. The working fluid was taken as ideal compressible air.

Because of the meshing considerations, the moving elements are not connected directly to their counterparts at the sealings, therefore here the flow is restricted by a porous domain defined within a narrow area limited to couple of cells within a given radius around the theoretical contact points. Here the porous resistance was also defined by Darcy-Forchheimer source term. In both cases, the viscous resistance was set to the maximum allowed value, for which the stability of the solution is ensured.

The models were tested at two distinct throttling configuration. The predicted variation of the pressure at the cylinder exit just upstream the porous zone is shown in Figures 5 (a) and (b). The results align relatively well in both low and high throttling cases. Although, despite that the same loading was imposed for both models, the predicted pressure ratio reached higher values in case of the redesigned and simplified model compressor (Fig. 2 (b)) than in case of the original design resembling the exact geometry of the Magai compressor (Fig. 2 (a)). Hence, the gradient of the curves representing the actual pressure rise in the model compressor is slightly higher, both in Fig. 5 (a) and (b).



(a)



(b)

Figure 5. Pressure difference between the inlet and the outlet normalized with the maximum value at (a) high and (b) low throttling conditions, predicted by the original model resembling the Magai type compressor geometry and the simplified model compressor

In case of the model compressor, the predicted volumetric efficiency was 93% at lower throttling and 90% at higher throttling. On the other hand, the same values are consecutively 88% and 84% in case of the original Magai type compressor model. The discrepancies are the result of the higher leakage flow predicted by the original model. In the original geometry there are two extra sealings connected to the vane which were not taken into the account in the simplified model. However, the leakage flow can be adjusted by changing the viscous resistance of the porous zone at the sealing contact positions, and part of the vane blade can be turned to be permeable to simulate the leakage across the vane. Still, there is not much chance to significantly improve the predictions before relevant measurement data are available for appropriate tuning.

4. COMPRESSOR AND DISCHARGE VALVE

Rolling piston compressors are usually implemented with a discharge valve to prevent back flow when the discharge port is connected to a high pressure reservoir. The applied valves are usually simple reed valves made of high-grade steel. However, the Magai type compressor was planned to be coupled with a commercially available pneumatic check valve for the duration of the first test period because of practical reasons. The numerical model is presented in Figure 6.

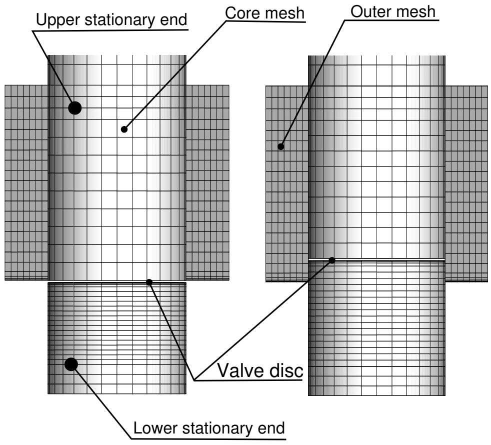


Figure 6. The numerical model of the discharge valve in closed and fully opened position

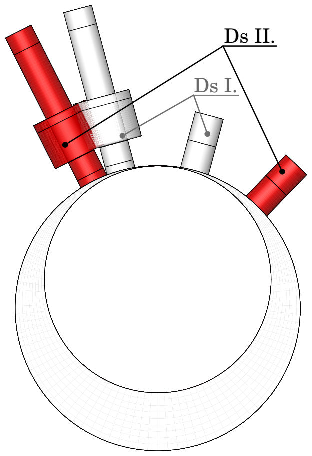


Figure 7. Representation of the different setups

The core and the outer part of the mesh is connected across a non-conformal mesh interface. The core part between the upper and lower stationary end is dynamic and follows the movement of the pressure activated valve disc. The position of the node points within this dynamic region is controlled by ANSYS Fluent's smoothing algorithm and no re-meshing was applied. This solution can be only used with the simplified compressor model, since the smoothing algorithm is not accessible when the 2.5D model is active. Because of the vane design of the Magai type compressor, the tangential distance between inlet and the outlet ports is relatively high compared to the traditional rolling piston type compressor architectures. This can result in decreased volumetric efficiency,

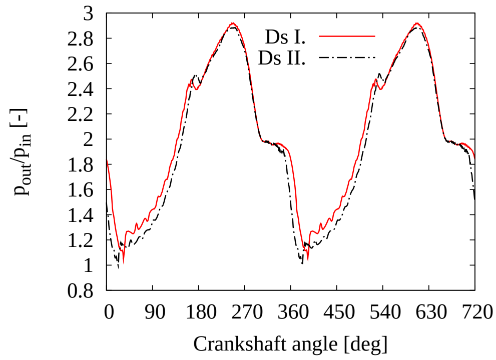


Figure 8. Ratio between the inlet and the cylinder outlet pressure in function of the Crankshaft angle in case Ds I. and Ds II.

since the effective compressed volume is reduced. To investigate the effect of the tangential separation of the inlet and outlet ports, two distinct designs were tested as it is shown in Figure 7. In both cases loading was imposed by the increased pressure at the outlet boundary, which resembles a scenario when the outlet port is connected to a pressurized system. This ratio was set to be relatively low and the pressure at the outlet boundary is twice as high as the inlet pressure. Figure 8 show the variation of the pressure ratio between the cylinder outlet pressure upstream the discharge valve and the inlet ambient pressure. There is no significant difference between the two curves. However, in case of DsI., when the in-take and the discharge ports are closer to each other, the pressurization starts earlier and the discharge process takes longer. There is a distinct overshoot in maximum pressure compared to the outlet pressure at the outlet boundary, which is the result of the throttling imposed by the sudden area change between the cylinder and the discharge pipe, and also of the additional throttling imposed by the discharge valve itself. The

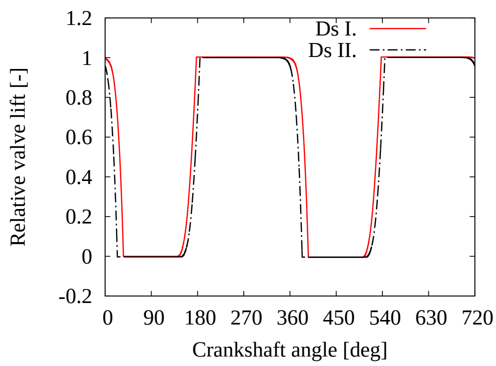


Figure 9. The relative valve lift in function of the Crankshaft angle in case Ds I. and Ds II.

relative valve lift is presented in Figure 9. The area ratio between the outlet port and free surface of the opened valve is 0.95. The variation of the valve lift also confirms the longer discharge process in case of Ds I., although no significant difference can be observed. However the predicted volumetric efficiency is dropped from 94% to 86% as the angular distance was increased between the inlet and outlet ports from Ds I. to Ds II..

5. CONCLUSION

To improve the performance of the traditional rolling piston type compressors, a new design was introduced. To verify the expectations regarding the redesigned system the CFD model of the new geometry was prepared and tested.

To increase the computational efficiency and reduce computational, original model was simplified and a new 3D moving mesh approach was introduced.

The new method was compared to the original algorithm which was provided by the ANSYS Fluent solver.

Despite some marginal discrepancies, the viability of the new method was proven although no final conclusion can be withdrawn before sufficient experimental data will be available.

The new moving mesh approach has proved the potential for further development and the possibility to adapt it for more complex geometries. Finally, a more complex compressor model coupled with a discharge valve was introduced to investigate the effect of the increased angular separation between the suction and the discharge port.

It was estimated that it has moderate effect on the overall performance at low pressure ratios. If these trends can be experimentally confirmed than a future parametric study can be conducted with a new model to find an optimal design.

6. ACKNOWLEDGEMENT

This research has been supported by the New Széchenyi Plan under contract No. KMR_12-1-2012-0199.

REFERENCES

[1] I. Magai, "http:// www.magaimotor.magai.eu," 2014.

[2] ASHRAE, *Compressors*. 2000 ASHRAE Systems and Equipment Handbook, 2000.

[3] B. G. Prasad, "Cfd for positive displacement compressors," in *International Compressor Engineering Conference*, (Purdue University), July 12-15 2004.

[4] W. Geng, C. H. Liu, and Y. Z. Wang, "The performance optimization of rolling piston compressors based on cfd simulation," in *International Compressor Engineering Conference*, (Purdue University), July 12-15 2004.

[5] C. H. Liu and W. Geng, "Research on suction performance of two-cylinder rolling piston type rotary compressors based on cfd simulation," in *International Compressor Engineering Conference at Purdue*, (Purdue University), July 12-15 2004.

[6] S. Liang, S. Xia, X. Kang, P. Zhou, Q. Liu, and Y. Hu, "Investigation of refrigerant flow simulation and experiment of rolling piston," in *International Compressor Engineering Conference at Purdue*, (Purdue University), July 12-15 2010.

[7] H. Ding and H. Gao, "3-D transient cfd model for rolling piston compressor with dynamic reed valve," in *International Compressor Engineering Conference at Purdue*, (Purdue University), July 14-17 2014.

[8] R. D. Brancher and C. J. Deschamps, "Modeling of rolling-piston compressors with special attention to the suction and discharge processes," in *International Compressor Engineering Conference at Purdue*, July 14-17 2014.

[9] *ANSYS Fluent documentation., v14.5, http://www.ansys.com*, 2012.

[10] B. Farkas, V. Szente, and J. M. Suda, "A simplified modeling approach for rolling piston compressors," *Period. Polytech. Mech. Eng.*, vol. 59, no. 2, pp. 94–101, 2015.

[11] M. Okur and I. S. Akmandor, "Experimental investigation of hinged and spring loaded rolling piston compressors pertaining to a turbo rotary engine," *Applied Thermal Engineering*, vol. 31, no. 6-7, pp. 1031–1038, 2011.