ABSTRACT

In the Compressed Air Energy Storage (CAES) approach, air is compressed to high pressure, stored, and expanded to output work when needed. The temperature of air tends to rise during compression, and the rise in the air internal energy is wasted during the later storage period as the compressed air cools back to ambient temperature.

The present study focuses on designing an interrupted-plate heat exchanger used in a liquid-piston compression chamber for CAES. The exchanger features layers of thin plates stacked in an interrupted pattern. Twenty-seven exchangers featuring different combinations of shape parameters are analyzed. The exchangers are modeled as porous media. As such, for each exchanger shape, a Representative Elementary Volume (REV), which represents a unit cell of the exchanger, is developed. The flow through the REV is simulated with periodic velocity and thermal boundary conditions, using the commercial CFD software ANSYS FLUENT. Simulations of the REVs for the various exchangers characterize the various shape parameter effects on values of pressure drop and heat transfer coefficient between solid surfaces and fluid. For an experimental validation of the numerical solution, two different exchanger models made by rapid prototyping, are tested for pressure drop and heat transfer. Good agreement is found between numerical and experimental results. Nusselt number vs. Reynolds number relations are developed on the basis of pore size and on hydraulic diameter.

To analyze performance of exchangers with different shapes, a simplified zero-dimensional thermodynamic model for the compression chamber with the inserted heat exchange elements is developed. This model, valuable for system optimization and control simulations, is a set of ordinary differential equations. They are solved numerically for each exchanger insert shape to determine the geometries of best compression efficiency.

1. INTRODUCTION

Compressed Air Energy Storage (CAES) technique can be integrated into power generation plants to overcome the mismatch between power generation and power demand during the daily cycles of a wind turbine, for instance. In the CAES approach, air is compressed to high pressure during low electric demand periods, stored, and expanded to output work during high electric demand periods [1], [2]. The compression operates in cycles; a compression stroke usually takes seconds. During compression, the air temperature tends to rise, and thus part of the input work is converted into the rise of internal energy of air. At the end of each compression, the compressed air is released into a storage vessel from the compression chamber. The rise of the internal energy due to the compression process is wasted during storage as the compressed air cools. Thus, enhancing heat transfer during compression will reduce temperature rise, and, consequently, reduce the amount of work input for the same pressure compression ratio, leading to
improving compression efficiency. The present study focuses on designing an interrupted-plate heat exchanger used in a liquid-piston compression chamber of the CAES system for the purpose of reducing the temperature rise during compression. The interrupted-plate heat exchanger is of the regenerative-heat-exchanger type. In most applications, regenerative heat exchangers operate in cycles; each cycle consists of a cooling process, in which heat is transferred from a hot fluid to the exchanger, and a heating process, in which heat is transferred from the exchanger to a cold fluid. In the CAES, compression and expansion strokes do not operate alternately; compression is followed by exhaust of compressed air when energy is being stored and expansion is followed by intake of compressed air when power is generated from compressed air. The present study will investigate heat transfer of the exchanger for the purpose of cooling the air during compression.

In a previous study [3], [4], a microfabricated segmented-involute-foil regenerator was designed for a Stirling engine. It successfully enhanced heat transfer, due to the stacked interrupted foils that force creation of new thermal boundary layers as flow passes through the foil layers. In the present study, the same idea of generating new thermal boundary layers is used; hence, an interrupted-plate heat exchanger is designed.

In the present study, the fluid flow and heat transfer are analyzed through CFD simulations. Because of its repeating geometry, the heat exchanger is analyzed as a porous medium. In the literature [5] – [9], the flow-resistance characteristics of porous media have been modeled using a Darcian and a Forchheimer extension term; the interfacial heat transfer between a porous medium and its surrounding fluid has been modeled using Nusselt number and Reynolds number correlations. If the exact geometry of the porous medium is known, these models can be obtained computationally from CFD simulations on a Representative Elementary Volume (REV) of the porous medium [10] and [11].

To quantify the exchanger performance, a method involving the number of pressure heads (NPH) and the number of transfer units (NTU) has been proposed [12]. The strength of this method is that it captures the trade-off between pressure drop and heat transfer of heat exchangers. The weakness, though, is that it puts equal weights on the importance of pressure drop and heat transfer. In the CAES application, due to compression, the input work needed to overcome the thermodynamic pressure may significantly outweigh the pressure drop of flow through the exchanger. Thus the NPH and NTU method may not be completely suitable for CAES.

Simple, transient one-dimensional regenerative heat exchanger models have been developed to give quick and accurate solutions for heat transfer and fluid temperature in exchangers [13] – [15]. In these models, the gas is not compressed within the exchanger. Also, a zero-dimensional model has been solved for air being compressed in a long, thin tube section of an air compressor [16]. In the present study, a zero-dimensional compression model featuring two energy equations, one for the air, and one for the solid, is developed. The solution of the model is used to estimate the compression efficiency, a parameter that quantifies CAES performance [17], [18].

2. EXCHANGER GEOMETRY AND PARAMETERS

The interrupted-plate heat exchanger is made of layers of stacked plates. The adjacent layers are perpendicular to one another. A schematic of the exchanger is shown in Fig. 1 (a). A schematic of an REV, representing a unit cell of the exchanger, is shown in Fig. 1 (b). The periodicity of the exchanger structure is the same in the y and z directions. Three dimensions of the exchanger are: plate length, \( l \), plate thickness, \( t \), and plate distance, \( 2b \). Twenty-seven shapes are analyzed with various combinations of \( l \), \( t \), and \( 2b \). The list of parameters for the various shapes is shown in Table 1.

![Flow direction](image)

(a) Schematic of the exchanger for a cylindrical chamber

![Flow direction](image)

(b) Schematic of an REV of the exchanger

Fig. 1. Interrupted-plate exchanger shape and parameters

<table>
<thead>
<tr>
<th>Cases</th>
<th>REV1</th>
<th>REV2</th>
<th>REV3</th>
<th>REV4</th>
<th>REV5</th>
<th>REV6</th>
<th>REV7</th>
</tr>
</thead>
<tbody>
<tr>
<td>( l ) (mm)</td>
<td>7.5</td>
<td>7.5</td>
<td>15</td>
<td>15</td>
<td>7.5</td>
<td>7.5</td>
<td>7.5</td>
</tr>
<tr>
<td>( t ) (mm)</td>
<td>0.4</td>
<td>0.8</td>
<td>0.4</td>
<td>0.8</td>
<td>0.4</td>
<td>0.8</td>
<td>1.6</td>
</tr>
<tr>
<td>( 2b ) (mm)</td>
<td>5</td>
<td>10</td>
<td>5</td>
<td>10</td>
<td>2.5</td>
<td>5</td>
<td>10</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cases</th>
<th>REV8</th>
<th>REV9</th>
<th>REV10</th>
<th>REV11</th>
<th>REV12</th>
<th>REV13</th>
<th>REV14</th>
</tr>
</thead>
<tbody>
<tr>
<td>( l ) (mm)</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>7.5</td>
<td>15</td>
<td>15</td>
<td>15</td>
</tr>
<tr>
<td>( t ) (mm)</td>
<td>0.4</td>
<td>0.8</td>
<td>1.6</td>
<td>0.8</td>
<td>1.6</td>
<td>0.4</td>
<td>1.6</td>
</tr>
<tr>
<td>( 2b ) (mm)</td>
<td>2.5</td>
<td>5</td>
<td>10</td>
<td>2.5</td>
<td>5</td>
<td>2.5</td>
<td>5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cases</th>
<th>REV15</th>
<th>REV16</th>
<th>REV17</th>
<th>REV18</th>
<th>REV19</th>
<th>REV20</th>
<th>REV21</th>
</tr>
</thead>
<tbody>
<tr>
<td>( l ) (mm)</td>
<td>4</td>
<td>4</td>
<td>2</td>
<td>4</td>
<td>4</td>
<td>3</td>
<td>2</td>
</tr>
<tr>
<td>( t ) (mm)</td>
<td>0.8</td>
<td>0.4</td>
<td>0.4</td>
<td>1.6</td>
<td>0.6</td>
<td>0.4</td>
<td>0.8</td>
</tr>
<tr>
<td>( 2b ) (mm)</td>
<td>2.5</td>
<td>2.5</td>
<td>2.5</td>
<td>10</td>
<td>2.5</td>
<td>2.5</td>
<td>5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cases</th>
<th>REV22</th>
<th>REV23</th>
<th>REV24</th>
<th>REV25</th>
<th>REV26</th>
<th>REV27</th>
</tr>
</thead>
<tbody>
<tr>
<td>( l ) (mm)</td>
<td>3</td>
<td>2</td>
<td>5.5</td>
<td>5.5</td>
<td>5.5</td>
<td>5.5</td>
</tr>
<tr>
<td>( t ) (mm)</td>
<td>0.8</td>
<td>1.6</td>
<td>0.8</td>
<td>0.8</td>
<td>0.4</td>
<td>0.4</td>
</tr>
<tr>
<td>( 2b ) (mm)</td>
<td>5</td>
<td>10</td>
<td>5</td>
<td>2.5</td>
<td>5</td>
<td>2.5</td>
</tr>
</tbody>
</table>

3. SOLUTION PROCEDURE

3.1. Governing Equations

Simulations by CFD on the REV models of different shapes are conducted. The goal is to calculate pressure drop and heat transfer with uniform and constant plate temperature, and use these to estimate the exchanger performance. The computational
The domain is the fluid region of the REV model, the transparent region shown in Fig. 1(b). The continuity, momentum, and energy equations are solved. The Reynolds number based on the plate length is defined as,

\[ Re_l = \frac{\rho l u_{REV,f}}{\mu_{REV,f}} \quad (1) \]

Where \( u_{REV,f} \) is the volume-averaged fluid velocity in an REV, and \( \mu_{REV,f} \) is the mean viscosity in an REV. Based on flow regime criteria given in [19], for simulations with \( Re_l < 400 \), the laminar flow formulation is solved. For the CFD simulations with \( Re_l = 400 \), the \( k-e \) model based on [20] is used. The governing equations for turbulent flow simulations, as well as physical properties, and turbulence model coefficients are given in Table 2. Default model coefficients based on [20] are used.

Table 2. Governing Equations for the RANS Calculations (fluid variables are time-averaged; upper bars are neglected), Physical Properties, and Turbulence Model Coefficients

\[ \begin{align*}
\text{Continuity:} & \quad \frac{\partial \bar{u}_i}{\partial x_i} = 0 \\
\text{Momentum:} & \quad \frac{\partial \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu + \nu_t \right) \left( 2S_{ij} - \frac{2}{3} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right) - \frac{\partial}{\partial x_i} \left( 2 \frac{k}{\rho} \frac{\partial \bar{u}_j}{\partial x_j} \right) \\
\text{k equation:} & \quad \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_i} \right) + \nu_t (2S_{ij}S_{ij}) - \epsilon \\
\text{\( \epsilon \) equation:} & \quad \frac{\partial \epsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (\nu + \nu_t) \frac{\partial \epsilon}{\partial x_i} \right] + C_1 \nu_t (2S_{ij}S_{ij}) \frac{1}{2} \epsilon - C_2 \frac{\epsilon^2}{k} + (\nu e) \frac{1}{2} \\
\text{Energy:} & \quad \frac{\partial}{\partial x_i} \left[ \bar{u}_i (\rho c_p T + p) \right] = \left[ \frac{k}{\rho R_T} \frac{\partial T}{\partial x_i} \right] + \nu_t \left( 2S_{ij} - \frac{2}{3} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right) \\
\text{Physical Properties:} & \quad \rho = 1.205 \text{kg/m}^3, \quad \mu = 1.716 \left( \frac{T}{273K} \right)^{5/3} \times 10^{-5} \text{kg/s}, \quad \kappa = 0.0257 \text{W/°C}, \quad c_p = 1006 \text{J/kgK} \\
\text{Turbulence Model Coefficients:} & \quad C_\mu = 0.09, \quad C_1 = 1.44, \quad C_2 = 1.92, \quad P_r = 0.85, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3
\end{align*} \]

3.2. Boundary Conditions and CFD Runs

The coordinate system is given in Fig. 1(b). The origin is at the center of the REV model. On the solid-fluid interfaces, no-slip velocity condition, and constant and uniform temperature at 293K are imposed. Periodic velocity and thermal boundaries are imposed on the entering-flow boundary and on the exiting-flow boundary,

\[ \Gamma|_{x=-l} = \Gamma|_{x=l}, \quad \Gamma = \bar{v}, \bar{k}, \epsilon, \theta \quad (2) \]

\( \theta \) is a dimensionless temperature based on bulk flow temperature and wall temperature,

\[ \theta = \frac{T - T_{bulk}}{T - T_0} \quad (3) \]

\[ T_{bulk}|_{x=l} = \int_{x=l} \frac{\rho c_p h T dA}{\int_{x=l} \rho c_p h dA} \quad (4) \]

\[ T_0 = 293K \quad T_{bulk}|_{x=-l} = 303K \]

The remaining boundaries are symmetric,

\[ \left( \frac{\partial \bar{u}}{\partial y} \right)_{y=\pm(b+\frac{t}{2})} = 0, \quad \chi = \bar{v}, \bar{k}, \epsilon, T \]

\[ \left( \frac{\partial \bar{u}}{\partial z} \right)_{z=\pm(b+\frac{t}{2})} = 0, \quad \chi = \bar{v}, \bar{k}, \epsilon, T \quad (5) \]

Another imposed condition is the mass flow rate,

\[ m = Re_l \mu A_{cross,f} \quad (6) \]

\( A_{cross,f} \) is the cross sectional area of the volume occupied by the fluid phase in an REV.

A total number of 58 CFD runs were computed. Two different Reynolds numbers, \( Re_l = 1 \) and \( Re_l = 400 \), are simulated for each shape listed in Table 1. In addition, four CFD runs are computed for the following cases: REV21 with \( Re_l = 9.4 \), REV23 with \( Re_l = 13.1 \), REV22 with \( Re_l = 81.7 \), and REV11 with \( Re_l = 363.1 \). In each simulation, the mass flow rate is specified according to Eq. (7). The direction of the mass flow is along the \( x \) axis.

3.3. Numerical Scheme and Verification

The transport equations are solved by the finite volume method, using the commercial CFD software ANSYS FLUENT. The Green-Gauss Cell Based method from [21] is used for the gradient formulations. The SIMPLE algorithm [22] is used for pressure-velocity coupling. The pressure values at cell faces are obtained by interpolation based on momentum calculations, a method introduced in [23]. The momentum and energy equations are discretized using the 2nd-order upwind method; the turbulence kinetic energy (\( k \)) and dissipation rate (\( \epsilon \)) equations are discretized using the 1st-order upwind method. The convergence criteria for residuals of all equations (10^-6) is set.

The computational domain is discretized into rectangular hexahedra cells. The cell size gradually decreases as the walls are approached; the maximum edge size of cells is four times the minimum edge size of cells. Different cell sizes are used for different REV models due to different geometry and size. Among the twenty-seven REV models, the number of computation cells varies from 251,712 (case REV5) to 1,850,688 (case REV10), while the number of computation cells per volume varies from 532/mm^3 (case REV10) to 7658/mm^3 (case REV17).

For grid-independence verification, the case with the smallest amount of cells and the case with the smallest cells per volume are simulated with finer meshes, at \( Re_l = 400 \). Comparisons between the original cases and the grid-independence verification cases are shown in Table 3.
Cases with name extension ".i" represent verification cases corresponding to the original cases. It shows that a total number of 251,712 cells and a number of 532 cells per mm³ are sufficient. In the current study, all simulations are done with a grid cell number and a grid cells per volume density larger than these values.

<table>
<thead>
<tr>
<th>Cases</th>
<th>Number of Cells</th>
<th>Cells per Volume (mm³)</th>
<th>Pressure Drop (Pa/m)</th>
<th>Mean Fluid Temp. T (K)</th>
<th>Mean Wall Heat Transfer Coefficient h(W/m²K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>REV5</td>
<td>251,712</td>
<td>2315</td>
<td>-56.135</td>
<td>298.88</td>
<td>75.25</td>
</tr>
<tr>
<td>REV5_i</td>
<td>788,640</td>
<td>7252</td>
<td>-54.743</td>
<td>298.84</td>
<td>73.20</td>
</tr>
<tr>
<td>REV10</td>
<td>1,850,688</td>
<td>532</td>
<td>-2.724</td>
<td>300.30</td>
<td>24.28</td>
</tr>
<tr>
<td>REV10_i</td>
<td>7,098,224</td>
<td>2040</td>
<td>-2.621</td>
<td>300.23</td>
<td>23.28</td>
</tr>
</tbody>
</table>

4. CFD RESULTS

4.1. Pressure Drop Calculations and Experimental Validation

The interrupted-plate exchanger is considered to be a porous medium. The pressure drop is modeled using a Darcian and a Forchheimer extension term,

\[ \frac{dp}{dx} = -\frac{\mu}{K} u_d - \frac{F \rho u_d^2}{K^2} u_d \]  \hspace{1cm} (8)

For each shape, the permeability, \( K \), is calculated using the pressure drop value computed from the \( Re_l = 1 \) run, and the Forchheimer coefficient, \( F \), is calculated using the pressure drop value computed from the \( Re_l = 400 \) run. The Darcian velocity, \( u_d \), by definition, is the fluid velocity averaged over the total REV area. Since the mass flow rate for each simulation case is known, the Darcian velocity can be obtained by

\[ u_d = \frac{m}{\rho A_{cross,t}} \]  \hspace{1cm} (9)

\( A_{cross,t} \) is the total cross-sectional area of an REV. The permeability and the Forchheimer coefficient are calculated for each shape, and are used later for the analysis of compression efficiency (see Section 5).

Two interrupted-plate models (REV6 and REV11) are fabricated using rapid prototyping. One model is made for REV6, and two are made for REV11 (namely REV11_1 and REV11_2). A picture of them is shown in Fig. 2. The interrupted-plate models are made by two different 3-D rapid prototype printers, model REV6 and model REV11_1 by Dimension stl1200es, and REV11_2 by Stratasys Print Plus. Model REV11_2 and model REV6 are used for pressure drop measurements to compare with the CFD calculations. Model REV11_1 is used in a compression experiment that will be shown in Section 5.

A schematic of the experimental setup for measuring pressure drop of the interrupted-plate exchanger models is shown in Fig. 3. A fan withdraws air from the right side to the left side as shown in the figure. The fabricated interrupted-plate model is inserted in the pipe, at the right end. The flow rate of air can be adjusted by the manual valve. A Sierra Top Track 822S flow meter is used to measure the volumetric flow rate. The gauge pressure is measured from the pressure tap by a micro-manometer, and is considered as the pressure drop of the interrupted plate. The measurement is taken at the Turbulent Convective Heat Transfer Lab at the University of Minnesota.

![Fig. 2. Fabricated exchanger models](Image)

![Fig. 3. Schematic of experimental setup for measuring pressure drop of the interrupted-plate model](Image)

![Fig. 4. Pressure Drop Comparison, Experiment vs. CFD](Image)

Due to fabrication variation, model REV6 has a plate thickness of 0.6mm and Model REV11_2 has a plate thickness of 0.9mm, while the design thickness is 0.8mm for both. For the purpose of comparison, CFD simulations are done by changing the REV geometries to the exact geometries of those fabricated models for the experiment. Based on the simulations, the computed permeability and Forchheimer coefficients for experimental models REV6 and REV11 are calculated. Equation (8) is then used to calculate the pressure drop, and the results are shown in Fig. 4, compared with the results from experimental measurements. The uncertainty associated with the pressure measurement by the micro-manometer is 0.125Pa. The relative uncertainty associated with the measured Darcian velocity is 1.5%, given by the flow meter manufacturer. Overall agreement is found between CFD and experiment.

4.2. Heat Transfer

Two heat transfer coefficients are calculated based on CFD results. The mean wall heat transfer coefficient, \( h \), measures the amount of heat transfer per solid surface area; the volumetric heat transfer coefficient, \( h_v \), measures the heat transfer per exchanger volume. They are defined as:
where $\bar{q}$ is the area-averaged wall heat flux, $T_{bulk,f}$ is the mass-averaged fluid temperature in an REV. The volumetric heat transfer coefficient directly shows the exchanger’s heat transfer capability when it is used in a compression chamber with a fixed volume. It is plotted against plate length and thickness in the three plots in Fig. 5, each plot featuring a fixed plate distance.

From Fig. 5, for a fixed plate distance, the volumetric heat transfer coefficient increases as the plate length decreases. The plate thickness has almost no effect on heat transfer for a given plate length and distance, for the range investigated (0.4mm < t < 1.6mm). Changing the plate length has a significant effect on heat transfer, especially when the plate length is less than 5mm. Comparing the three graphs in Fig. 5 shows that decreasing the plate distance improves heat transfer. Among all exchanger shapes, the one with the largest volumetric heat transfer coefficient (122kW/(m²K)) is REV17, which is the one with the smallest values of all three dimensional parameters. The one with the smallest volumetric heat transfer coefficient (4.5kW/(m²K)) is REV10, which is the one with the largest values of all three dimensions.

The flow fields for REV17 and REV10 are shown in Fig. 6. In addition to the fluid velocity and temperature fields, the wall heat flux on the solid surface is shown in each plot. A small stagnation region is shown near the frontal area of the plate due to direct impact with the flow. The maximum local heat flux is found at the edges of the frontal area. The flow is effectively cooled by the interrupted plates. The hottest spot of the flow is in the core of the fluid region.

Two methods of defining the dimensionless numbers are used to present the results. The first one is based on hydraulic diameter, and the second one is based on pore size [9]. For the flow direction investigated in the present study, the hydraulic diameter of the interrupted-plate channel is,

$$D_h = 4b$$

The Reynolds number based on hydraulic diameter is:

$$Re_D = \frac{\rho \bar{V} D_h}{\mu}$$

No explicit way of calculating the pore size for geometries like the interrupted-plate exchanger has been found in the literature. In the present study, it is considered that one REV has the equivalent of two pores. Therefore, the pore size is estimated by:

$$L = \frac{3}{\sqrt[3]{\epsilon V_{REV}}/2}$$

The Reynolds number based on pore size is:

$$Re_L = \frac{\rho \bar{V} D_h}{\mu}$$

Two Nusselt numbers are defined as:

$$Nu_{Dh} = \frac{h_D}{\kappa}$$

$$Nu_V = \frac{h_V l^2}{\kappa}$$

From a total of fifty-eight CFD runs, $Re_D$, $Nu_{Dh}$, $Re_L$, and $Nu_V$ are calculated and are plotted in Fig. 7.

The flow through interrupted plates possesses features of both channel flow and flat plate flow. In Fig. 7 (a), the data cover three different hydraulic diameters, 5mm, 10mm and 20mm. When $Re_D$ is less than 100, the flow is laminar-like and $Nu_{Dh}$ is nearly a constant; when $Re_D$ becomes larger than 100, $Nu_{Dh}$ increases with $Re_D$. These features have some similarity to the situation of flow in a channel. In Fig. 7 (b), the dimensionless numbers are based on pore size, L. The pore sizes vary from case to case; they cover the range between 12.0mm (REV10) and 2.44mm (REV17). In Fig. 7 (b), the change in $Re_L$ is primarily affected by the change in the pore size. When $Re_L$ is between 0.2 and 3, $Nu_L$ decreases as $Re_L$ increases, similar to laminar flow over a flat plate with growing boundary layers. When $Re_L$ is between 3 and 20, $Nu_L$ is generally not affected by $Re_L$, similar to a steady laminar flow. When $Re_L$ increases from 60 to 150, $Nu_L$ has a significant increase, followed by a region of scattered results where $Re_L$ is between 150 and 500. This feature is similar to laminar-to-turbulent transition. Overall agreement is found between Fig. 7 (b) and the criteria of porous media flow regimes given in [19]. These criteria are shown in Table 4.
Porous media heat transfer correlations are usually given in the following general form:

\[ Nu = a + b \cdot Re^c Pr^d \]  \hspace{1cm} (18)

Data obtained from the simulation runs suggest the following correlation using a least square fit:

\[ Nu_{dh} = 9.700 + 0.0876 Re_{dh}^{0.792} Pr^{0.3} \]  \hspace{1cm} (19)

The correlation is plotted together with the computed data from the CFD runs, as shown in Fig. 7 (a).
5. Zero-D COMPRESSION MODEL AND EFFICIENCY ANALYSIS

5.1. Formulation and Numerical Approach

The primary application of the interrupted-plate heat exchanger in the present study is for a cylindrical, liquid-piston compression chamber for CAES. The liquid-piston approach injects liquid (water, in the present study) into a compression chamber, and uses the liquid column as a piston to compress air in the chamber. To quickly compare performance of using different exchangers, a zero-D model representing the compression process in a chamber with an inserted exchanger is solved. The model is based on the first law of thermodynamics, using the heat transfer results obtained from the previous CFD simulations. For the air phase, the energy conservation gives,

\[ Q = mc_v \frac{dT_a}{dt} + pV \frac{dV}{dt} \]

(20)

After substituting the ideal gas law for the temperature term, and using a constant liquid piston velocity for the volume flow rate term, and assuming heat transfer is only between the air and the exchanger insert, Eq.(20) becomes,

\[ \bar{\eta}_V V(T_a - T_s) = (\frac{c_v}{R} + 1) \varepsilon A U_{in} P + \frac{c_v}{R} \varepsilon A U_{in} V \frac{dP}{dV} \]

(21)

where \(A\) is the cross sectional area of the chamber. The energy conservation for the solid phase is:

\[ c_s \rho_s (1 - \varepsilon) AU_{in} \frac{dT_s}{dt} = -\bar{\eta}_V (T_a - T_s) \]

(22)

The ODE system formed by Eqns. (21) and (22) requires input of an averaged volumetric heat transfer coefficient, \(\bar{\eta}_V\).

\[ \bar{\eta}_V = a_V \bar{h} = a_V \frac{k \bar{N} u_{DH}}{D_h} \]

(23)

It is assumed that the instantaneous air velocity distribution is linear along the chamber axis, matching the liquid piston velocity at the piston end and zero at the cap end. Averaging:

\[ \bar{N} u_{DH} = 9.700 + 0.0876 \bar{R} e_{DH} 0.792 P r ^{-1} \]

\[ = 9.700 + 0.0489 (\frac{\rho U_{in} D_h}{\mu}) 0.792 P r ^{-1} \]

(24)

Equations (21) and (22) are solved by the finite difference method, using an iterative implicit scheme. Equation (21) is discretized by:

\[ \bar{\eta}_V^{(i)} V T_s^{(i+1)} - T_s^{(i)} = (\frac{c_v}{R} + 1) \varepsilon AU_{in} P^{(i+1)} + \frac{c_v}{R} \varepsilon AU_{in} V T_s^{(i+1)} - P^{(i)} \]

(25)

Superscript \((n)\) represents the volume marching step, and superscript \((i)\) represents numerical iteration step within one volume step. During each iteration, \(P^{(i+1)}\) is solved from Eq.(25), and then \(T_s^{(i+1)}\) is solved from:

\[ c_s \rho_s (1 - \varepsilon) AU_{in} \frac{T_s^{(i+1)} - T_s^{(i)}}{\Delta V} = -\bar{\eta}_V^{(i)} (\frac{V^{(n+1)} + V^{(n)}}{mR} P^{(i+1)} - P^{(i)}) \]

(26)

Next, the index of iteration step, \(i\), increases by 1 and Eqns.(25) and (26) are solved again. Once convergence of this iterative process is reached, \(P^{(i+1)} = P^{(i+1)}\), and \(T_s^{(n+1)} = T_s^{(i+1)}\). Then the computation marches to the next volume step, by increasing the index of volume step, \(n\), by 1.

5.2. Experimental Validation of the zero-D Model

Experimental measurements of liquid-piston compression with insertion of the interrupted-plate model are completed and the results were compared with computed results using the zero-D model. The experimental setup is shown in Fig. 8. A water pump provides the required flow rate to compress air inside the compressor chamber. As shown in Fig. 8, water is driven from left to right, through the relief valve, the Coriolis flow meter, the turbine meter, the control valve, and into the compression chamber, which is the transparent tube on the right. The compression chamber wall is made of acrylic, which has a thermal conductivity between 0.17W/(mk) and 0.2W/(mk), and a heat capacity of 1470J/(kgK). It is assumed that heat transfer effects from air to the chamber wall, and from air to the water are both negligible, compared to heat transfer from air to the exchanger. Two pressure transducers are mounted in the setup to measure the upstream water pressure and the downstream air pressure. An Omega FTB-1412 turbine flow meter and an EndressHauser Promass 80 Coriolis flow-meter are used to measure the water volume and mass flow rate, respectively. The Coriolis meter measurement has a time lag of 71ms, and the turbine meter is not accurate at small flow rates. Therefore, these two meters are used together to obtain accurate measurement of the volume flow of water into the compression chamber, and consequently the volume of the air under compression. The relief valve is mounted in the circuit to keep the pressure of the circuit below 160 psi. Filters are also included to remove unwanted particles from the flowing water. A 16-bit PCI-DAS1602/16 multifunction analog and digital I/O board is used to acquire the system data. This board is linked with MATLAB/Simulink via xPC Target to send and receive the experimental measurements of liquid.

An experiment is run with the interrupted-plate model REV11_1 (Fig. 2). The compression chamber has a length of 35cm and diameter of 5.08cm, and, thus, a volume of 709.4cm³. Water is driven at a near-constant flow rate to compress air. The average flow rate is 2.69 x 10^-4 m³/s, equivalent to an inlet water velocity of 0.159m/s. Due to the nature of the control valve, the flow rate is not constant at the very beginning and the very end of compression (referring to Fig. 9). The total length of
the exchanger insert is 30cm, and it is inserted to fill up the entire upper part of the compression chamber. In the lower part, there is a region of 5cm length that does not have insert material. In the experiment, compression starts with the bottom of the chamber. The compression time in the experiment is 2s and the pressure compression ratio is 10. The initial temperature in the experiment is 298K. Application of the experiment in a real power plant utilizes much larger compression chambers and hydraulic motors, which can have a storage power of 5MW, by compressing 4144kg air to 200 times atmospheric pressure in 5min.

![Experimental setup using liquid-piston compression for CAES at the Fluid Power Control Lab at the University of Minnesota](image)

Fig. 8. Experimental setup using liquid-piston compression for CAES at the Fluid Power Control Lab at the University of Minnesota

The zero-D model is solved to simulate the experimental run. The simulation starts at the time when water enters the bottom of the exchanger region. The fabricated exchanger model REV11_1 has a hydraulic diameter of 5.5mm, a specific surface area of 643/in, and a porosity of 0.833. These are used in the zero-D model, with the heat transfer correlation, Eq. (24). The physical properties of air are the same as listed in Table 2, except that density is not constant. Physical properties of the solid are: \( \rho_s = 1060 \text{kg/m}^3 \) and \( c_s = 1200/(\text{kgK}) \).

Comparisons between the zero-D model solutions and the experimental results are shown in Fig. 10. The spatial pressure variation in the chamber is assumed negligible. The instantaneous volume, pressure and temperature are non-dimensionalized based on their initial values. In the experiment, the uncertainty of the pressure transducer is 0.1%; the uncertainty in the transient volume measurement is \( 3 \times 10^{-6} \text{m}^3 \). The temperature is calculated from ideal gas law, based on pressure and volume measurements. The uncertainties of the initial volume and initial temperature measurements are respectively \( 1 \times 10^{-6} \text{m}^3 \) and 0.8 K. Using an uncertainty propagation equation from [24], the uncertainty of the transient temperature measurement is 4.39%, where the percentage is based on absolute temperature. In the experiment, the maximum temperature rise during the compression process is 47K, which has an uncertainty of 15K. This is a high fractional value of the temperature rise simply because the matrix is so successful in keeping the temperature rise small. The cumulative quantities have higher percentage accuracy. Using the same uncertainty propagation equation, the cumulative heat flow to the matrix in the experiment is 118.2J, with an uncertainty of 2.52J, which accounts for a relative uncertainty of 4.44%. Overall agreement is found between the zero-D model and experimental results by comparing pressure and temperature trajectories, as shown in Fig. 10. In the second half of the compression process, there is disagreement between temperatures calculated from the zero-D model and those of the experiment. A possible reason for this is that the zero-D model does not start the simulation from the beginning of compression in the experiment. At the end of compression in the experiment, the temperature drops. This is caused by the slow-down of the compression speed due to the nature of the valve; the zero-D model does not have this feature.

![Instantaneous Air Volume during Compression in Experiment](image)

Fig. 9. Instantaneous Air Volume during Compression in Experiment

(a) Dimensionless pressure (b) Dimensionless temperature

Fig. 10. Comparison between zero-D model and experiment

5.3. Efficiency Analysis

The performance values of the 27 exchanger shapes are evaluated based on compression efficiency, a merit parameter used in the CAES application [2], [17], [18]. In the CAES approach, air is compressed to a high pressure, stored, and expanded to output work when needed. The storage energy is defined as the amount of work extraction from the compressed air as it would undergo an isothermal expansion process to the atmospheric pressure:

\[
E_s = mR_T_0(ln\zeta - 1 + \frac{1}{\zeta})
\]  

(27)

The work input to compress air takes place in two phases. In the first phase, compression work is done to compress the air from atmospheric pressure to a high pressure. In the second phase, which is the post-compression storage period, the air cools. In order to maintain the work potential (storage energy) of the stored, compressed air as it cools, additional work is done on the air as its volume decreases, such that its storage pressure is maintained. This source of additional work input is identified as cooling work. The total work input is the sum of the compression and the cooling work. Thus, the efficiency of compression in CAES is defined as the storage energy divided by the total work input:

\[
\eta = \frac{E_s}{W_{in}} = \frac{mR_T_0(ln\zeta - 1 + \frac{1}{\zeta})}{-\int_{V_0}^{V_1} [(P-tot-P_0) dV + (P_0-P_d)(V-V_0)]}
\]  

(28)

The total pressure needed to drive the water to compress air is the sum of the thermodynamic pressure of air and the pressure
drop as the water passes through the exchanger:
\[ P_{\text{tot}} = P + P_r = P + \left( \frac{\mu L}{K} U_{\text{in}} + \frac{P_r U_{\text{in}}}{R_d^2} \right) \mathcal{L} \]  
(29)

Where \( \mathcal{L} \) is the instantaneous length of the portion of the exchanger that is immersed in the water column.

The efficiency, given by Eq. (28) is a merit parameter to quantify performance of different exchanger shapes in the present study. In order to calculate the efficiency of the exchanger in the compression chamber, the zero-D model is used, along with the heat transfer correlation, Eq. (19), to solve for the pressure rise trajectory during compression for each exchanger shape, by substituting the corresponding shape parameters, and the corresponding permeability and Forchheimer coefficients into the model. The compressor in all zero-D model simulations features a length of 0.3m and a constant compression speed of 0.2m/s. The fluid and solid properties are the same as those used in Sections 3.1 and 5.2. Physical properties of water (the liquid piston) are: \( \rho_w = 998 \text{ kg/m}^3 \) and \( \mu_w = 1.31 \times 10^{-3} \text{ kg/(ms)} \).

After solving the zero-D model for each exchanger shape, the compression efficiency is evaluated for each exchanger shape for a pressure compression ratio of 10. The results are shown in Fig. 11. For a given plate distance, decreasing the plate length helps improve efficiency. The effect of plate thickness on efficiency is small for the range investigated. In general, increasing it results in a slight increase in efficiency, for a fixed plate distance and plate length. Comparing the three graphs in Fig. 11, one sees that decreasing the plate distance leads to improvement of efficiency. Among all the exchanger shapes investigated, the best two are: REV17 (\( l = 2\text{mm}, t = 0.4\text{mm}, 2b = 2.5\text{mm}, \eta = 0.921 \)) and REV15 (\( l = 4\text{mm}, t = 0.8\text{mm}, 2b = 2.5\text{mm}, \eta = 0.920 \)).

![Fig.11. Compression efficiency of different exchanger shapes, pressure compression ratio 10](image)

6. CONCLUSIONS
The present study has investigated interrupted-plate heat exchangers of twenty-seven different shapes through CFD simulations on the REV models. Heat transfer results obtained from the CFD calculations are presented in terms of dimensionless parameters. The curve of Nusselt number vs. Reynolds number based on pores size shows similar features to a developing, laminar flow over flat plate, transitioning into a turbulent flow. The curve of Nusselt number vs. Reynolds number based on the hydraulic diameter shows similar features to a channel flow. A heat transfer correlation is obtained by fitting the data.

The exchanger performance is analyzed based on compression efficiency, a merit parameter for CAES. In order to calculate the compression efficiency, a simple zero-D compression model formulated on the heat transfer results from the REV simulations, is proposed and validated by an experiment. The study shows that, in general, decreasing the plate length or plate distance helps improve the efficiency. Plots of efficiency vs. different shape parameters have been obtained.

**NOMENCLATURE**

- \( a_f \) Specific area of porous medium
- \( h \) Half distance between adjacent plates
- \( c_p \) Constant-pressure specific heat
- \( c_v \) Constant-volume specific heat
- \( D_h \) Hydraulic diameter
- \( E_\text{s} \) Storage energy of CAES
- \( F \) Forchheimer coefficient
- \( \bar{h} \) Area-averaged surface heat transfer coefficient
- \( \bar{h}_v \) Volumetric heat transfer coefficient
- \( \bar{h}_{h} \) Average volumetric heat transfer coefficient for zero-D model
- \( K \) Permeability
- \( k \) Turbulence kinetic energy
- \( \kappa \) Thermal conductivity of air
- \( L \) Pore size
- \( l \) Plate length
- \( \mathcal{L} \) Instantaneous length of compression chamber
- \( \dot{m} \) Mass flow rate
- \( \lambda_{h} \) Nusselt number based on \( D_h \)
- \( \lambda_{\dot{m}} \) Nusselt number based on \( h_v \) and \( L \)
- \( \bar{\lambda}_{h} \) Average hydraulic-diameter Nusselt number in zero-D model
- \( P \) Air pressure in the zero-D model
- \( P_r \) Pressure drop due to exchanger insert
- \( Pr \) Prandtl number
- \( p \) Local pressure
\( Q \) Instantaneous heat flow rate from the air to the exchanger
\( \bar{q} \) Area-averaged heat flux
\( R \) Ideal gas constant of air
\( Re_{dh} \) Reynolds number based on \( D_h \)
\( Re_L \) Reynolds number based on \( L \)
\( Re_t \) Reynolds number based on \( l \)
\( S_{ij} \) Strain rate using tensor index notation
\( T \) Local fluid temperature
\( T_0 \) Initial temperature; wall temperature
\( T_a \) Air temperature in zero-D model
\( T_{bulk} \) Local fluid bulk temperature
\( T_{bulk,f} \) Mass-averaged fluid temperature in an REV
\( T_s \) Solid temperature in zero-D model
\( t \) Plate thickness
\( t \) Time
\( U_{in} \) Liquid piston velocity
\( u \) Local x-component velocity
\( u_i \) Velocity using tensor index notation
\( V \) Instantaneous volume of chamber
\( W_{in} \) Work input
\( \bar{v} \) Local velocity vector
\( x, y, z \) Cartesian coordinates
\( \bar{x}_j \) Cartesian coordinates using tensor index notation

Greek Symbols
\( \epsilon \) Turbulence dissipation
\( \varepsilon \) Porosity
\( \eta \) Compression efficiency
\( \mu \) Dynamic viscosity
\( \nu \) Kinematic viscosity
\( \nu_t \) Turbulence kinematic viscosity
\( \rho \) Density
\( \theta \) Local dimensionless fluid temperature
\( \zeta \) Final pressure compression ratio

Subscripts
0 Initial value of variable
\( a \) Air
\( d \) Darcian velocity
\( f \) Values at the end of compression
\( REV, f \) Volume-averaged on the fluid volume in an REV
\( s \) Solid
\( tot \) Total
\( w \) Water

Superscripts
(i) Numerical iteration step
(n) Numerical volume marching step
* dimensionless variable

ACKNOWLEDGEMENTS
The authors would like to thank our collaborators at the University of Virginia, and in particular, Jackson Cheek and Prof. Eric Loth, for fabricating one of the rapid-prototype exchanger models. This work is supported by the National Science Foundation under grant NSF-EFRI #1038294, and University of Minnesota, Institute for Renewable Energy and Environment (IREE) under grant: RS-0027-11. The authors would like to thank also the Minnesota Super-Computing Institute for the computational resources used in this work.

REFERENCES


