Clearance Analysis and Leakage Flow CFD Model of a Two-Lobe Multi-Recompression Heater

Ashish M. Joshi,¹ David I. Blekhman,² James D. Felske,³ John A. Lordi,³ and Joseph C. Mollendorf³

¹ CD-adapco, 9401 General Drive, Plymouth, MI 48170, USA

² Department of Mechanical Engineering, School of Engineering, Grand Valley State University, Grand Rapids, MI 49504, USA

³ Department of Mechanical and Aerospace Engineering, State University of New York at Buffalo, Buffalo, NY 14260, USA

Received 21 February 2006; Revised 5 May 2006; Accepted 4 June 2006

This paper reports the results of a study on multi-recompression heating. This process employs a Roots-type mechanism to heat gases to very high temperatures by compressive gas heating. A CFD model predicting the leakage flows in the machine was developed, and an excellent comparison with experimental data taken on a two-lobe Roots blower was obtained. A “clearance analysis” was performed to show that the clearance between the impellers remains constant for 96% of the angles of rotation. Assuming a quasi-steady state, the CFD simulation was performed for a single angle of rotation. A three-dimensional analysis showed that the flow field is identical along the rotor length, except for the leakage through the end plates. Hence, the model was further simplified to a two-dimensional analysis. This research may provide guidance in predicting the leakage flows in other blowers of the same kind with a different geometry.

Copyright © 2006 Ashish M. Joshi et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

1. INTRODUCTION

The temperature rise due to compression, although considered as undesirable in pumping applications, can be harnessed to produce an uncontaminated high-temperature (1500 K or higher), uniformly heated stream of gas. Many industrial applications require a clean, spatially uniform, high-temperature gas stream. Such applications are found in pyrolytic processes chemistry for waste conversion and green chemistry, as well as in industrial processes such as fiber optic production. The best presently known methods to heat gas at a high-temperature are direct electrical resistance heating and combustion. Current technology used to heat gas does not provide a clean gas stream, nor does the stream spatially uniform in temperature. This results in disadvantages for various applications. One solution to this is compressive gas heating, which is volumetrically uniform and does not require chemical reactions to heat the gas. The multi-recompression heater (MRH) is a mechanical type heater, embodied as a modified Roots compressor, which works on the principle of compressive heating through multiple compressions of gas to create a high-temperature gas stream.

One of the major advantages of a two-lobe Roots compressor is its high volumetric displacement rate. An experimental study of compressive gas heating was performed by Blekhman [1] on a Whispair 404J RAM Roots two-lobe compressor. As the MRH compresses the gas, a high-pressure reservoir is developed on the discharge side. Lubrication-free operation of the Roots compressor comes at the price of having clearances which allow for pressurized gas to escape to the inlet region. This pressurized gas is at high-temperature and preheats the incoming gas and causes it to expand. This reduces the volumetric efficiency of the machine.

In the current research, a robust “computational model” is developed to predict the leakage flows in the machine. Mathematical equations for the geometry of the impeller were derived. A rotor-rotor clearance analysis was performed to show that the clearance remains constant for about 96% of the angles of rotation. Assuming a quasi-steady state, the CFD simulation was performed. Computational results showed good comparison with experimental results.

As the objective of this research was to study only the leakage flow in the multi-recompression heater, an unsteady three-dimensional moving mesh analysis was not performed. The ongoing research focuses on the unsteady effects, which

coupled with this study will be a valuable tool for designing similar Roots blowers.

2. BACKGROUND

The Roots compressor operates on the same compression principle as the multi-recompression heater (see Figure 1). The Roots compressor consists of two impellers on parallel shafts enclosed in a housing in which the inlet and discharge ports are located. The shafts on which the impellers are mounted are equipped with bearings on both ends and with timing gears on one or both ends, comprising the external drive train of the compressor. One of the shafts is driven by an appropriate power transmitting mechanism which can be directly coupled to a motor or through a drive belt. Half the torque to the drive shaft is transmitted to the second shaft by the timing gear. Each impeller has two lobes. There are clearances between two impellers (rotor-rotor clearance) and clearance between impellers and casing (rotor-casing clearance). This allows for lubrication-free operation. Modern manufacturing techniques can produce very tight clearances. As the rotors start rotating, the gas enters through the inlet port. During the rotation of the impellers, the gas is trapped in the “well.” As soon as the impeller passes the breakaway point, the gas in the well mixes with the gas in the reservoir downstream. After multiple rotations, pressure starts building up in the outlet region and a high-pressure reservoir is developed. This causes “leakage” gas to flow back into the low-pressure region through the clearances. The leaked gas, being at a high-temperature, heats the incoming gas. The volume of the incoming gas increases, decreasing the volumetric efficiency and hence reducing the mass flow rate significantly. The presently reported research is to study this leakage flow in detail, with the aid of computational fluid dynamics.

3. IMPELLER GEOMETRY

The 404J RAM Roots compressor has a two-lobe design for the impellers. The two-lobe design has a high volumetric displacement. Figure 2 shows the actual shape of the impeller in the Roots compressor, called the “design shape” (solid line). Another shape called the “base shape” is defined which has a circle-involute-circle profile (dashed line). Mathematical equations are first derived for the “base shape” and are then modified to arrive at the equations of the “design shape.” The effect of the tip strip will be discussed later.

3.1. Base shape

If we consider the first quadrant (see Figure 3), the involute profile is a combination of three arcs: the mouth (or waist) and upper portion of the lobe are circular (R_1 and R_2) and the convex arc connecting them is an involute, which is formed by tracing the end of the string unrolling from the base circle (R_b).

The current profile is created with the following parametric equations for the waist, involute, and upper lobe shapes.

FIGURE 1: Schematic of a two-lobe Roots-type multi-recompression heater.

FIGURE 2: Base shape and design shape.

FIGURE 3: Profile of base shape.

For $\theta = [0, \beta_1]$,

$$\begin{aligned} x_1 &= R_1 \cos \left(\pi - \frac{5\theta}{4} \right) + x_0, \\ y_1 &= R_1 \sin \left(\pi - \frac{5\theta}{4} \right). \end{aligned} \quad (1)$$

For $\theta = [\beta_1, \beta_2]$,

$$\begin{aligned} x_1(\theta) &= R_{21}(\psi) \cos(-\beta_1) + R_{22}(\psi) \sin(-\beta_1), \\ y_1(\theta) &= R_{22}(\psi) \cos(-\beta_1) - R_{21}(\psi) \sin(-\beta_1), \end{aligned} \quad (2)$$

where

$$\begin{aligned} R_{21}(\psi) &= R_b \{ \cos \psi + \psi \cdot \sin \psi \}, \\ R_{22}(\psi) &= R_b \{ \sin \psi - \psi \cdot \cos \psi \}. \end{aligned} \quad (3)$$

The angle “ ψ ” goes from 0° to 86.4708° when θ goes from 40° to 70° . The relation between ψ and θ is as follows:

$$\theta = \tan^{-1} \left[\frac{\sin \psi - \psi \cdot \cos \psi}{\cos \psi + \psi \cdot \sin \psi} \right]. \quad (4)$$

For $\theta = [\beta_2, \pi/2]$,

$$\begin{aligned} x_1(\theta) &= R_3 \cos \left[\frac{52.6\pi(\theta - \beta_2)}{180(\pi/2 - \beta_2)} + \frac{37.4\pi}{180} \right], \\ y_1(\theta) &= R_3 \cdot \sin \left[\frac{52.6\pi(\theta - \beta_2)}{180(\pi/2 - \beta_2)} + \frac{37.4\pi}{180} \right] + y_0. \end{aligned} \quad (5)$$

The values shown in Figure 3 are as follows (inches for linear dimensions and radians for angles): $R_1 = 1.44$, $R_b = 0.8 * R_p = 1.787$, $R_3 = 1.37$, $x_0 = 2.255$, $y_0 = R_p = 2.239$, $\beta_1 = 4\pi/18$, and $\beta_2 = 7\pi/18$. By substituting the above values in (1)–(5), it is found that the lobe is 3.119 inches wide, as shown in Figure 3.

3.2. Design shape

The design shape is a modification of the base shape. To add the “add-ons” to the involute curve, the normal to the involute curve has to be found. To find the normal to the involute at any point, the gradient (slope) of the normal is first found out by taking negative reciprocal of dy/dx ,

$$\frac{-1}{dy/dx} = m = \frac{\sin \psi \cdot \sin \beta_1 - \cos \psi \cdot \cos \beta_1}{\sin \psi \cdot \cos \beta_1 + \cos \psi \cdot \sin \beta_1}. \quad (6)$$

The thickness of add-ons is “ a ” = 0.04115 in. Hence, the extra material added will be

$$\begin{aligned} X_{\text{add}} &= a \cdot \cos(\tan^{-1} m), \\ Y_{\text{add}} &= a \cdot \sin(\tan^{-1} m). \end{aligned} \quad (7)$$

We add this to the x and y coordinates of the “base shape.” Hence,

$$\begin{aligned} x_{\text{design}} &= x_1(\theta) + X_{\text{add}}, \\ y_{\text{design}} &= y_1(\theta) + Y_{\text{add}}. \end{aligned} \quad (8)$$

FIGURE 4: Rotor-rotor clearance analysis.

For the “tip strip” a value of 0.055 is added for $\theta = 89.444^\circ$ to 90° . This is the angle where the tip strip starts and is obtained from the machine drawing of the “design shape.”

4. CLEARANCE ANALYSIS

Since the Roots blower has two-lobe symmetrical impellers, the rotor positions repeat themselves after 180° , and the clearance data repeats itself after 90° and is symmetrical about 45° . The coordinates of rotors at any angle of rotation are found by a combined “rotational/translational” coordinate transformation. To find the minimum clearance as a function of rotation angle from 0° to 45° , only single quadrants of rotor #1 and of rotor #2 need to be considered, as shown in Figure 4.

To find the minimum clearance, the distance between each point on rotor #1 and each point on rotor #2 is found. This is done with the aid of two nested “for loops.” The minimum of all the distances is then the minimum clearance, and the two points where it occurs, defines its position. With the above algorithm, the minimum clearance between the rotors was found from 0° to 45° with an increment of 0.5° . As one would expect, the exact minimum clearance will be obtained when there are infinite number of grid points on the impeller curve and the increments in the angle of rotation are infinitesimally small. But for the purpose of numerical modeling, it is found that, with $n = 1440$, the oscillations in the clearance data with $n = 1440$ are of the order of 10^{-5} inches.

A “clearance analysis” was performed for both the “base shape” and the “design shape.” The graph of clearance versus angle of rotation for the “base shape” and “design shape” is shown in Figure 5. It is seen that the addition of the extra material on the involute section results in a constant clearance for a larger part of the cycle (from 1.5° – 45°) as compared to the “base shape” (from 9.5° – 45°). Also, the clearance for the “design shape” is much smaller as compared to the “base shape.”

5. CFD MODEL

The computational fluid dynamics study was performed using a commercial CFD code (Star-CD).

FIGURE 5: Minimum clearance with base shape and design shape.

5.1. Assumptions

After steady state has been reached, rotation of the impellers has little or no influence on the pressure ratio across the machine. Thus, a quasi-steady state is assumed. The CFD simulation for the full machine is performed only for the base shape and the effect of the tip strip on the rotor-casing leakage path is studied separately. Also, the flow field is assumed to be the same at any cross-section along the rotor length except for the leakage through the end plate clearances. Hence, a two-dimensional analysis was performed and the results for the flow rate were simply scaled with the rotor length to get the total leakage.

5.2. Turbulence model

The flow exiting the gap experiences a strong adverse pressure gradient (discussed later). The $k - \omega$ turbulence model was therefore adopted since it gives accurate results for wall-bounded flows with adverse pressure gradients. The model provides two additional equations for k and ω . More details on the $k - \omega$ turbulence model may be found in [2].

5.3. Spatial discretization

Star-CD uses a finite-volume method to discretize the differential equations of mass, momentum, energy, k , and ω . The use of higher-order schemes is often a balance between accuracy and ancillary requirements such as resistance to spurious numerical oscillations and ease of solution. The numerical oscillations, or “wiggles,” usually arise when the ratio of convection to diffusion in a cell is large, and this ratio is usually characterized by the mesh-Peclet number or mesh-Reynolds number. Lower-order schemes produce “numerical diffusion,” that is, smearing of gradients. This is a form of truncation error that diminishes as the grid is refined, but at an increased cost of computation. Amongst the

various higher order schemes available in Star-CD, the linear upwind differencing (LUD) scheme was found to perform the best. Other higher-order schemes such as the central differencing scheme, quick scheme (quadratic upstream interpolation of convective kinematics), and SFCD scheme (self-filtered central differencing) were found to exhibit non-physical spatial oscillations. These oscillations were caused by equations, which were difficult to solve because of a large stencil size. They also produced negative densities and negative turbulent kinetic energy.

The linear upwind differencing scheme is specially formulated for nonstructured meshes and derived from a scheme originally proposed for structured meshes [3]. This higher-order upwind differencing scheme approximates the first derivative term (convective term) by fitting a parabola to the point and its two upwind neighbors. It is conventional in the sense that only function values are used; it does not give rise to wiggles and has only a slight tendency for overshoots; it is $O(h^2)$.

This scheme yields.

For $u \geq 0$,

$$u \frac{\partial \phi}{\partial x} \cong \frac{3u}{2h} (\phi_i) - \frac{2u}{h} (\phi_{i-1}) + \frac{u}{2h} (\phi_{i-2}). \quad (9)$$

The right-hand side of the above equation can be reduced to a system of tridiagonal equations, which are diagonally dominant and converge rapidly. Similarly, an expression for $u \leq 0$ can be written.

5.4. Solution algorithms

Star-CD employs implicit methods to solve the algebraic finite-volume equations resulting from discretization. This procedure iterates to the steady state and eliminates the temporal derivatives which give rise to stability issues. The algorithm used to solve the set of finite-volume equations is called SIMPISO, which combines the elements of the SIMPLE and PISO algorithms. The SIMPISO algorithm is used solely for steady-state calculations in an iterative mode.

All the above algorithms share the following features [4].

- (i) They employ the predictor corrector strategy, (enabled by the use of operator splitting), to temporarily decouple the flow equations so that they can be solved sequentially.
- (ii) Continuity is enforced with the aid of an equation set for pressure, derived by combining the finite-volume momentum and mass conservation equations.
- (iii) The solution sequence involves a predictor stage, which produces a provisional velocity field derived from the momentum equations and a provisional pressure distribution. The provisional fields are then refined in the corrector stages until both the momentum and continuity balances are satisfied to some approximation.
- (iv) For iterative calculations, the above step is repetitively executed until the solution is reached. Underrelaxation is used to stabilize the process. Thus lower values of “ α ”

are used in the following:

$$\phi^n = a \cdot \phi^n + (1 - a) \cdot \phi^{n-1}. \quad (10)$$

- (v) With the sequence, the operator split equation sets involve only one of the field variables.

The SIMPISO scheme borrows from the PISO scheme, a more elaborate treatment of the pressure gradient terms arising from grid nonorthogonality. It is more robust for distorted meshes. It employs another relaxation factor (set to 0.6) as it requires additional work per iteration.

5.5. Solution convergence

To determine if the solution has converged, the residuals at each time step are monitored. The residual r_ϕ of the finite-volume solution at a particular cell and iteration n is the imbalance of the FV transport equation:

$$r_\phi^n \equiv A_p \phi_p^k - \sum_m A_m \phi_m^k - s_1. \quad (11)$$

The measure actually employed to judge the convergence is the normalized absolute residual sum:

$$R_\phi^k \equiv \frac{\sum |r_\phi^k|}{M_\phi}, \quad (12)$$

where the summation is over all the cells in the domain and M_ϕ is a normalization factor. For this research, convergence was determined by using a residual tolerance of 1.000E-06.

5.6. Mesh generation

The geometry was modeled in Pro/Engineer and imported into Star-CD as an IGES file. Alternatively, it was created from the vertices (obtained while performing the “clearance analysis”) by using pro-STAR (Star-CD preprocessor) commands. The model, when imported from the IGES file, is stored in Star-CD in the form of “splines.” The size and shape of the cells play an important role in numerical accuracy and stability. The smaller the cells, the more the accuracy, and more the computational time. Very small cells, that is, a fine mesh, are not needed where the gradients in the flow are small. But a very fine mesh has to be constructed in the regions of high-pressure, velocity, or temperature gradients. This corresponds to the clearance regions in the current research. Also, boundary layer effects are important in these regions. The “shape” of the cell is described by its aspect ratio, internal angle, and warp angle. An ideal two-dimensional cell is square with no warpage. Deviation from this results in adverse effects such as negative densities. The solution becomes less stable, and might not converge. Figure 6 shows the mesh.

The fine mesh in the rotor-rotor clearance is shown in Figure 7. The y^+ values for the cell centroids near the wall boundaries are of the order of 1.0. The aspect ratios for the two-dimensional mesh (shell cells) were maintained below 8.0. As the code is a finite-volume code, the 2D mesh is extruded one-cell thick. The thickness of the cells (and the

FIGURE 6: Mesh for the two-dimensional simulation.

FIGURE 7: Mesh in rotor-rotor gap.

3D aspect ratio) is arbitrary, as “symmetry boundaries” are formed on either side.

A discussion of the different types of boundaries is given in the next section. Blocks of disconnected cells, that is, cells with different mesh densities, are connected by “couples” in Star-CD [4]. Figure 7 shows a couple of joining cells with different mesh densities.

5.7. Boundary conditions

As described earlier, the CFD model is a quasisteady model of the leakage flow in the machine. The actual flow will not be considered. Leakage flow originates from the high-pressure side and leaves the computational domain on the low-pressure side. In the actual machine, though, it is carried over to the high-pressure side by the rotation of the impellers before it can exit through the low-pressure side. Hence, the boundary conditions of the leakage model are flow coming in through the high-pressure side (model inlet), and going out through the low-pressure side (model outlet). Also, to solve a 2D flow, symmetry boundary conditions have to be imposed on the planes parallel to the Z-plane. This is based on the assumption that there is no velocity component in the Z direction. The symmetry boundary eliminates the consideration

of the 3D aspect ratio, and hence the single layer of cells can be assigned an arbitrary thickness while extruding. The unspecified boundaries are automatically assigned an “impermeable wall” boundary condition.

The experimental procedure [1] prescribes lowering the pressure on the actual inlet side by changing the position of a valve, and keeping the pressure on the actual outlet side as atmospheric ($1.013E + 05$ Pa) until the desired pressure ratio and/or temperature ratio is obtained. The experimental data provides the temperature ratio T_{out}/T_{in} for a given pressure ratio P_{out}/P_{in} . The inlet temperature of the gas is specified as room temperature (293 K).

To effectively simulate the experimental procedure, the pressure of the “model inlet” has to be kept constant and the pressure ratio adjusted by lowering the pressure at the “model outlet.” The computation is started by specifying the density, inlet velocities, static temperature, and turbulence parameters at the “model inlet,” and the static pressure and turbulence parameters at the “model outlet.” When the solution starts converging, the “model inlet” boundary conditions are changed to stagnation quantities, that is, stagnation pressure and temperature. The pressure in the interior is extrapolated to the boundary plane; the velocity at the boundary is then computed from the specified P_{stag} and the extrapolated P_{stat} and applied as a boundary condition. Then the momentum and mass equations are solved in succession. This switch in boundary conditions is necessary, as often-times the computation fails because the extrapolated P_{stat} exceeds P_{stag} .

Note in Figure 6 that the “model outlet” boundary is a converging channel. In Star-CD, large outflow pressure boundaries [4] often cause flow reversals due to local pressure gradients, and may lead to divergence due to negative densities near the boundary. When the flow comes in through outlet boundaries, the user-specified values for turbulence and temperature are applied causing local regions of large/small temperatures and turbulent viscosities. To avoid divergence, the flow has to be forced out of the domain through the outlet boundary. Hence, the outflow boundary was reduced to half its size by tapering, thereby accelerating the flow. Reducing the size of the outlet does not affect the quantity of interest, that is, leakage flow. The study of boundary conditions was done to simulate the experiments as closely as possible, within the quasi-steady nature of the CFD model.

6. RESULTS AND DISCUSSION

6.1. 2D results at 45° angle of rotation

After determining the effects of the tip strip and add-ons on the clearances and flow rates, a 2D simulation was performed for the full machine.

The case study is performed at the 45° angle, because the clearance remains constant for the “design shape” for more than 96% of the cycle, repeats itself after 90°, and is symmetrical about 45° as shown earlier. Case studies are done for the following rotor-rotor and rotor-casing clearance

FIGURE 8: Mach number contours for full machine.

combinations (RR = 0.014" & RC = 0.006", RR = 0.014" & RC = 0.011", and RR = 0.013" & RC = 0.014"). Rotor-case clearances vary in practice due to the tip strip manufacturing tolerances of $\pm 0.005''$ as well as wear during operation. The CFD calculations are performed for 4 pressure ratios (1.24, 2.09, 3.11, and 4.18). The corresponding temperature ratios are obtained from the experimental results of Blekhman [1].

To visualize the flow field, the Mach number, static pressure, and temperature plots are presented for a pressure ratio of 3.11, RR clearance = 0.013", and RC clearance of 0.014". Figure 8 shows the Mach number distribution for the entire flow field.

In general, it is seen that the flow velocities in the well regions are significantly smaller than in the gaps. The rotor-rotor and rotor-casing clearances create flow passages similar to converging diverging nozzles. Figure 9 is an expanded view of the Mach number contours in the rotor-rotor gap. It is observed that for a pressure ratio = 3.11, all gaps experience choked flow with the flow being sonic at the throat, then supersonic, and finally decelerating to subsonic speeds due to divergence of the nozzle area and friction.

The static pressure generally decreases through the machine. Figure 10 shows an expanded view of the pressure contours in the rotor-rotor flow passage. It can be seen that there is a gradual decrease in the pressure through the throat until the flow reaches its maximum Mach number. This is consistent with the flow in a subsonic-supersonic converging diverging nozzle [5]. As the nozzle continues to expand, the flow goes from supersonic to subsonic, with corresponding increase in the pressure. Then the pressure continues to

FIGURE 9: Mach number contours in rotor-rotor gap.

FIGURE 10: Static pressure contours for rotor-rotor gap.

increase and equalizes with the reservoir pressure. Thus, a $k - \omega$ turbulence model is proper for simulating the effects of this adverse pressure gradient. (In the case of an unchoked flow, the pressure also decreases until the throat and then increases, resulting in an unfavorable pressure gradient.)

FIGURE 11: Static temperature contours in rotor-rotor gap.

The static temperature field is uniform at the inflow boundary, which remains constant throughout the high-pressure reservoir. The static temperature decreases in the rotor-rotor and rotor-casing gaps, as shown in Figure 11 (rotor-rotor). It decreases gradually until the flow reaches the highest Mach number, and then increases to the temperature in the low-pressure outflow reservoir.

The velocity vector plot in Figure 12 shows the directionality of the flow through the different flow paths from the high-pressure reservoir to the low-pressure reservoir.

Flow recirculation is observed in the wells due to the high velocities near the casing which drag the neighboring fluid similar to a rotating flow in a cavity. It is observed that the recirculation regions are stronger in the model outlet reservoir because of the higher velocities.

6.2. Effects of the tip strip on rotor-casing clearance

The tip strip (Figure 13) has an interesting effect on the mass flow rate through the rotor-casing clearance. The tip strip with a 0.006'' clearance to the casing is shown on the right in Figure 13. The same 0.006'' clearance without the tip strip is shown on the left.

To study the effects of the tip strip on the rotor-casing leakage mass flow rates, the rotor-casing gap is simulated for several pressure ratios. The tangential velocity of the rotor at the highest point with the tip strip is ≈ 33 m/s, which gives a Mach number of ~ 0.1 . The direction of this velocity (V_T) is against the leakage flow as shown in Figure 13. This position

FIGURE 12: Velocity vector plots for entire machine.

FIGURE 13: Effect of tip strip on rotor-casing leakage.

is simulated by specifying an angular velocity for the rotor boundary with a no-slip condition. Except for this “rotating boundary,” all other boundaries and the mesh are kept stationary. The motivation is to determine the difference in flow rates between a rotating impeller with a tip strip and a nonrotating impeller without a tip strip.

It is observed that the leakage mass flow rates for the rotor-case gap with the tip strip are significantly lower for PRs less than 1.89, which is the PR for choking of the flow. After the flow chokes, the difference in mass flow rates is insignificant because Mach number is always equal to 1.0 near the throat. A comparison of leakage mass flow rates is shown in Figure 14.

Consequently, in order to effectively simulate the 404J Whispair Roots blower using the base shape, mass flow rates must be reduced by a factor proportional to the percent difference in the flow rates with and without the tip strip. This

FIGURE 14: Effect of tip strip and impeller rotation on rotor-casing leakage.

FIGURE 15: Mesh for 3D quasi-steady simulation.

factor is the percent contribution of the rotor-casing gaps towards the total leakage. Also, for $\pm 10^\circ$ around the 0° position, the tip strip creates a pressure drop in the rotor-rotor leakage path as well.

6.3. End plate leakage (quasi-steady, 3D)

The above analysis neglects the leakage through the end plate clearances. To estimate the effect of this leakage path, quasi-steady analysis was performed with a three-dimensional mesh as shown in Figure 15. The blue-colored cells are the end plate cells, one cell thick. The case study is performed with an angle of rotation of 45° and clearances of 0.011" for rotor-casing and 0.014" for rotor-rotor. The end plate clearance is specified as 0.005", which lies in the range specified by the Roots manual. The three-dimensional analysis is performed for 4 different pressure ratios. The results obtained are compared with the 2D mass flow rate results scaled with rotor length. The comparison shows that the end plate leakage is smaller for lower pressure ratios (4.15% for PR = 1.24) and increases with increasing pressure ratio (to up to 10.2% for PR = 4.18). These results are tabulated in Table 1.

FIGURE 16: Comparison of computational and experimental leakage for different clearance combinations.

TABLE 1: Comparison of 3D and 2D leakage mass flow rate results scaled with rotor length (RR = 0.014'', RC = 0.011'').

PR ratio	2D leakage scaled with rotor length (Kg/s) $\times 10^3$	3D leakage (quasi-steady) (Kg/s) $\times 10^3$	End plate leakage mass flow rate (Kg/s) $\times 10^3$
1.24	18.403	19.167	0.0764
2.09	17.632	18.671	1.0390
3.11	15.743	17.035	1.2920
4.18	14.693	16.194	1.5010

6.4. Resultant net leakage mass flow rate

The 2D leakage mass flow rates scaled with rotor length were reduced due to the effect of the tip strip and impeller rotation, and the leakage through the end plate clearance was added to compute the leakage under operating conditions.

Figure 16 compares computed and experimental leakage mass flow rates for different combinations of rotor-rotor and rotor-casing clearances. It is observed that the computational and experimental results have the same behavior, that is, leakage increases with increase in pressure ratio until PR = 1.89 and decreases thereafter for all clearance combinations. It is observed that the computational results for RR = 0.013'' and RC = 0.014'' match very well with the experimental results.

6.5. Conclusions and summary

A comprehensive clearance analysis was done of the two-lobe impeller geometry in the 404J Roots blower.

- (i) The clearance analysis shows that the material added to the basic involute shape gives a constant clearance for a larger part of the rotation (1.5°–90°). This simplifies the numerical procedure considerably, as the quasi-

steady analysis can now be performed with only one angle of rotation. This angle was chosen to be 45°, as the clearance data is symmetrical about this angle, and repeats itself after 90° of rotation.

- (ii) The tip strip is believed to be provided to reduce leakage as well as allow for wear and tear during the machine operation. The mass flow rates in the 2D computations for unchoked flow were less with the tip strip, since it creates a pressure drop in the rotor-casing clearance, which is significant for lower pressure ratios. It was also found that the tip strip reduces leakage for $\pm 10^\circ$ angles of rotation around the 0° position by creating a pressure drop in the rotor-rotor clearance.
- (iii) The three-dimensional quasi-steady analysis showed that the leakage due to the end plates was less (4%) for lower pressure ratios, and increases to up to 10% for higher pressure ratios.
- (iv) For the CFD analyses, a linear upwind differencing scheme suitable for elliptic problems was used, and found to give appropriate results. A $k - \omega$ turbulence model was used to capture the effects of the adverse pressure gradients in the flow exiting the gap. A very fine mesh with a $y^+ \sim 1$ for the first cell was used to make the mesh compatible to the turbulence model.

Considering all the effects of the clearances with special features on the design shape (tip strip and add-ons, end plate leakage), and physical effects of the impeller rotation, net leakage mass flow rates were obtained as a function of pressure ratio (Figure 16). It was observed that the computational and experimental results match well for RR = 0.013'' and RC = 0.014''. In addition to the analysis shown earlier, the tip strip is believed to further reduce the leakage mass flow rates for lower pressure ratios by pushing the fluid back into the high-pressure reservoir. This effect cannot be captured in a quasi-steady analysis in Star-CD. A definitive statement cannot be made unless an unsteady, moving mesh analysis is used to study this phenomenon. The influence of

the impeller rotation will probably be high due to the large surface area of the rotor, especially for lower pressure ratios. The effects of impeller rotation on the fluid between the end plates and rotor surfaces were not studied.

This research may be a valuable tool to design other blowers of the same kind but a different geometry as it gives a good approximation for the leakage flow in the machine, and hence the temperature rise of the incoming flow.

REFERENCES

- [1] D. I. Blekhman, "A theoretical and experimental study of high-temperature compressive gas heating," Dissertation, State University of New York at Buffalo, June 2003.
- [2] D. C. Wilcox, *Turbulence Modeling for CFD*, DCW Industries, La Canada, Calif, USA, 1998.
- [3] N. S. Wilkes and C. P. Thompson, "An evaluation of higher-order upwind differencing scheme for elliptic flow problems," in *Proceedings of the 3rd International Conference on Numerical Methods in Laminar and Turbulent Flow*, pp. 248–257, Seattle, Wash, USA, August 1983.
- [4] Star-CD methodology and user manual v3.15, CD-Adapco Group.
- [5] A. H. Shapiro, *The Dynamics and Thermodynamics of Compressible Fluid Flow*, Ronald Press, New York, NY, USA, May 1953.