A MULTI-PHASE CFD STUDY OF A LIQUID SLUG INGESTION IN A RECIPROCATING COMPRESSOR

Riccardo Traversari,
Rotating Machinery Design Manager
Compression Service Technology
Florence, Italy

Bernardo Bettini,
System Engineering Manager
Compression Service Technology
Florence, Italy

Cosimo Carasci,
Diagnostic Engineer
Compression Service Technology
Florence, Italy

Andrea Fusi
R&D Manager
Compression Service Technology
Florence, Italy

ABSTRACT

The fluid handled by reciprocating compressors has to be strictly limited to gas and, under no circumstances, can there be a liquid fraction ingestion, as this event can cause the famous “liquid hammer effect” that can generate catastrophic failures. A study was conducted to evaluate the quantitative effects of the ingestion of a liquid slug into a reciprocating compressor cylinder with the goal of calculating the pressure distribution on the piston during such an event. The Multi-phase CFD ANSYS FSI software, utilized to simulate the behavior of a mixture of gaseous and liquid hydrocarbons during the delivery phase, has shown that the pressure inside the cylinder increases up to five times the normal pressure. This new approach can be helpful in the design stage or for root cause failure analysis of large compressors such as the ones utilized in the natural gas, refining, and LDPE industries.

1. Introduction

It is well known that the fluid handled by reciprocating compressors has to be strictly limited to gas and, under no circumstances, should there be a liquid fraction ingestion as this event can cause the famous “liquid hammer effect” that can cause catastrophic failures of many compressor parts [1]. This is also explicitly stated in the API618 Standard governing the design and the operation of these machines [2]. Process compressors are typically part of complex plants, which contain equipment, such as reactors, heat exchangers, separators, etc., and it is very common, particularly in refineries and petrochemical plants, that the type of fluid treated is a mixture of hydrocarbons. The hydrocarbons may have a rather high dew point temperature while the plant layout, if not well designed and operated, can lead to uncontrolled cooling of process piping or to defective operation of the liquid fraction separators thus causing the abnormal condition of liquid ingestion. Similar events can occur in those plants where the gas is recycled and it entrains liquid fractions, such as waxes in polymerization processes, not removed by the process separator.

Figure 1: Schematic picture of a typical horizontal reciprocating compressor

The phenomenon of liquid inlet is as dangerous as insidious: beyond the extreme cases (continuous suction of only liquid with consequent stoppage or sudden destruction of the
compressor), sporadic incidents frequently occur in which one or more cylinders ingest significant liquid fractions for a certain number of cycles. The damage resulting from these events can seriously affect the life and safety of the machine even if no damage is apparent at the time of the event.

This article reports the theory and the results of a study that was conducted by the authors to quantitatively evaluate the effect of liquid ingestion into a reciprocating compressor cylinder designed for gaseous hydrocarbons. The goal was to calculate the pressure distribution on the piston during such an event.

2. Statement of the study

The thermo-fluid-dynamic phenomena that occur in a cycle of a reciprocating compressor, and in particular the pressure loss through the valves, ducts and manifolds, are a rather complicated event and were studied and modeled by many researchers. The investigation is even more difficult when it comes to two-phase fluids. They can be studied only by means of advanced simulation techniques.

The simulation in the study included the simulation of the arrival of a liquid slug at the intake of one end of a double acting cylinder, to determine the intensity of the forces that are generated in the various machine components. This study was conducted using the Multi-phase CFD FSI software simulating the behavior of a mixture of gaseous and liquid hydrocarbons during the delivery phase of that cylinder end.

2.1. Normal operation (gas)

Figure 2 shows a typical pattern of pressure in the cycle when the machine is handling gas in a normal running condition. For example, for the cylinder crank end, starting from the Bottom Dead Center (BDC), the steps are:

- Expansion of the gas in the clearance volume
- Suction up to Top Dead Center (TDC): the pressure inside the cylinder is below the nominal value of suction pressure due to the valve pressure drop
- Compression
- Delivery: the pressure inside the cylinder rises beyond the nominal value of the discharge pressure due to valve pressure drop.

The forces on the crank mechanism depend, other conditions remaining equal, on the maximum discharge pressure.

2.2. Operation in abnormal conditions (liquid ingestion)

If a cylinder is beginning to ingest a certain amount of liquid at every crankshaft revolution, one will find an increasing fraction of liquid in the cylinder during the subsequent phases of compression, in a way that is not simple nor unique to represent.

The diagrams of the discharge pressure will change dramatically and, as described in detail below, different models have been implemented (analytic zero-dimensional) that allow to estimate the liquid fraction inside the cylinder, the trend of pressure and the peak value reached. These models are highly sensitive to the values of the pressure loss factor assigned to the valve–valve port system.

2.3. Importance of transients

The pressure loss factors of valves are known with good approximation for the normal operation with gas in steady state (usually they are determined experimentally on test benches in steady state conditions). In the condition under investigation (liquid ingestion), one faces an operation with a gas-liquid mixture whose pressure loss factors are hardly predictable by analytical calculation methods. Of particular importance is the influence of the opening and closure transient of valves, covering a rate close to 35% of the duration of the whole delivery phase.

All this shows the inadequacy of calculations that use constant loss factors applied to such a transient phenomenon.

3. The method adopted for the multiphase analysis

We have observed how the nature of the physical phenomenon does not produce sufficiently accurate results with a zero-dimensional steady state approach. Also a CFD two-dimensional approach is inadequate for the problem complexity as it is not possible to identify symmetry conditions. The only adequate method to quantitatively analyze what happens in the cylinder and through the valves is a 3D CFD simulation.
Moreover, to achieve the aim of investigating the actions on the most critical parts of the machine, one needs to simulate the fluid dynamic transients and dynamic structural constraints that determine the flow and pressure behavior not only in terms of mean values but point by point and at every instant of time [3]. The method of investigation utilized by the authors consisted of the following sequence:

- Zero-dimensional analysis multi-phase involving more consecutives cycles;
- Determination of the initial conditions of the subsequent 3D CFD analysis;
- 3D CFD FSI Direct Analysis (movable meshes, rigid valve rings with rigid translation motion, multi-phase).

In order to validate the method, as it was practically impossible to make experimental measurements of the pressure inside the cylinder in this very special occurrence or reproduce artificially this exceptional situation, the algorithm was tested by calculating with this method the power adsorbed by the compressor and comparing it with the experimental measurements as well as comparing the calculated valve loss factor with data from the literature.

### 3.1. First step: Zero-dimensional approach

In order to keep the computing time of the subsequent CFD analysis within reasonable limits, a Zero-dimensional calculation algorithm was developed with Visual Basic code implemented in Excel to describe the pressure evolution in one end of the reciprocating compressor cylinder when, starting from the rated operating conditions with only gas, the liquid phase begins to enter the cylinder and determine the initial conditions of the subsequent compression and discharge phases.

In the case of the cylinder crank end, the calculation starts from the BDC of the piston with a zero amount of liquid inside the cylinder, only gas at discharge pressure within the clearance volume. The assumption for the calculation is that at the end of the expansion of the gas, the suction phase starts and only one liquid phase is sucked which gets mixed with the gas remaining in the clearance volume. The gas-liquid mixture is then compressed and discharged through the delivery valves. The following cycle starts with the clearance volume of the cylinder crank end containing not only gas but a mixture of gas and liquid.

The quantity of gas and liquid in the clearance volume conditions at the end of each cycle, are taken as initial conditions for the next cycle. The volume of liquid inside the cylinder increases during every subsequent cycle, since only fresh liquid is sucked, and a mixture of gas and liquid is discharged. In a couple of cycles the mixture of gas is strongly enriched in liquid phase.

The thermodynamic transformation is considered as isentropic.

### 3.2. Second step: The 3D CFD Model

The thermo-fluid-dynamic analysis is conducted using a fluid dynamic model that reproduces the cylinder crank end with the delivery valves and the inlet valves. The analysis is performed in transient and turbulent conditions, under the assumption of compressible mono or multiphase flow, and using deformable mesh calculation.

The deformation of the 3D domain changes the configuration of the cylinder, whose volume varies with a time law imposed by the law of motion of the piston and with the configuration of the valve, whose rings are moving in function of forces acting on them.

The mobile surfaces are those of the piston and valve rings within the valve. The simulation of opening and closure of the valve rings in CFD is obtained by solving the equation of the motion of the rings, treated as not deformable.

In the motion calculation, the dynamic forces acting on the rings, the characteristics of inertia and the forces due to the springs stiffness, have been taken into account. At this stage, it has been considered acceptable to treat the rings as infinitely rigid and to calculate a movement of pure translation, since the rings are fully open in the zone of the cycle where the maximum pressure occurs.

The software utilized for analysis are:

- ANSYS ICEM-CFD for the generation of geometry and mesh calculation
- ANSYS-CFX for fluid analysis

At the end of a functional construction of the calculation mesh and the definition of 3D regions with flexible walls, 4 different fluid domains were defined.

In the various regions of fluid, meshes with hexahedral elements are utilized, which allows better quality control of the meshes during the motion of the piston and rings, while tetrahedral prism mesh are used in those regions where deformations of the geometry do not occur. In fact a tetrahedral mesh is able to better describe in detail the geometric complexity of the areas close to the valve. The flow was considered transitional and turbulent (k-ε standard).

The resolution of the equation of total energy allowed to take into account the conditions of compressible flow. The action of gravity on the liquid phase was taken into account as its effect is not negligible.

In the multi-phase analysis, a homogeneous model with regard to velocity, temperature and turbulence was used. The use of a multi-phase model involves the solution of the transport equation for the variable "volume fraction". This variable allows the distribution of two phases in the system to be described.

The use of a homogeneous multi-phase model on a particular variable assumes that the two phases share the same field for the variable in question. With reference to the velocity, this means that at every point of the domain, gas and liquid are characterized by the same velocity vector. The basis of this model is the assumption that the exchange of momentum,
energy and turbulence between the two phases are sufficiently high to ensure that they are everywhere in equilibrium and therefore share the same fields. Under this hypothesis, it is possible to solve the transport equations using the properties of "bulk" that are calculated locally on the basis of physical properties and depending on the volume fractions of the two phases. Other conditions for the analysis are listed below:

- In the Multiphase analysis, liquid + gas are considered with immutable percentages, without the effects of evaporation or condensation.
- Mechanical parts are considered as undeformable (the profile of motion of the piston is known, and its deformation does not influence significantly the fluid dynamic field).
- Only one degree of freedom (axial translation) was assigned to the valve rings.
- The initial conditions for the 3D FSI analyses were derived from the zero-dimensional analysis method at the time of the compression start. It is interesting to note that the above model could be extended also to an FSI “2 way” analysis modeling the reciprocal interaction between the cylinder and the valves with the aim to study in detail also the valve movement. This option is not covered in this article.

Figure 3 summarizes the complete process both for the single-phase and for the multi-phase cases.

4. Application of the new method to a specific case

The machine considered for the study was a refinery compressor with the following characteristics:

- Compressor power: 400 kW
- Rotation speed: 500 RPM
- Inlet pressure: 15 bar a
- Normal delivery pressure: 21.2 bar a
- Gas: mixture of hydrocarbons with mol. weight of 11.7
- Density of the liquid phase: 600 kg/m3

4.1. Preliminary Zero-dimensional analysis

The zero-dimensional calculation was performed for four subsequent cycles of liquid ingestion. The results of this calculation are shown in Figure 4 where one can see that the pressure inside the cylinder reaches 116 bar-a just after the first liquid ingestion. The peak pressure arrives at 137 bar-a at the second ingestion cycle and achieves even higher values in the following revolutions. It is also interesting to note that the suction pressure, due to the much higher density of the liquid versus the gas one, decreases substantially in such a way that, when the piston is at the TDC, the value of the pressure is still very low and it takes a portion of the compression stroke to come back to the nominal suction pressure.

![Figure 4: Cylinder pressure during liquid ingestion cycle, calculated by zero-dimensional method](image)
approach taking the initial conditions determined with the zero-
dimensional calculation at the moment of closure of the suction
valve of the second cycle.
As previously explained, the initial conditions for this second
analysis were taken from the Zero-dimensional analysis and
were the following:
• Starting point: 251.5° crank angle from BDC
• Fluid pressure: 16.96 bar-a
• Volume of the liquid fraction: 9.60 dm³

4.2. Final 3D CFD-FSI direct analysis: results
The thermo–fluid–dynamic analysis was conducted using a
fluid dynamic model that reproduces a symmetrical portion of
the cylinder crank end with one delivery valve and one inlet
valve (Figure 5).
The results of the 3D CFD-FSI Multiphase analysis are
reported in Figure 7 from which one can see the fluid pressure
profile in four typical areas of the cylinder: the compression
chamber, the valve pocket, the ring area and the downstream
of the discharge valve. The peak pressure inside the cylinder
resulted 125 bar-a, against the value of 137 found with the
Zero-dimensional model and the nominal value with only gas
of 21.2 (see Figure 8).

Figure 6: reference regions for fig. 7 & 8 pressures

Figure 7: 3D CFD-FSI Multiphase analysis results

5. Validation of the model
The model was tested for validation by simulating the cylinder
behavior when running single phase with the specified gas. The
verification was done by comparing the results of the 3D CFD-
FSI model, for two parameters, with known data. The two
parameters were: the valve loss factor and the compressor
absorbed power. The valve loss factor was compared with the
value coming from the literature and from the experience of the
authors for the specific valve and valve port, while the absorbed
power was compared with measurements made in the field.
Figure 8 shows the results of the 3D CFD-FSI analysis run with
only gas in terms of pressure evolution in the points most
meaningful of the gas path.
The pressure difference between diagrams 1 (pressure inside
the cylinder bore) and 4 (pressure downstream the delivery
valve) allowed to calculate the effective valve loss factor, by
means of the equation:
\[ \xi P_f = 2 \Delta p / w^2 \rho \]

where:
- \( \Delta p \) = pressure loss at valves
- \( w \) = gas velocity
- \( \rho \) = gas density
- \( \xi \) = valve flow coefficient
- \( P_f \) = pocket factor.

By integrating this equation, the expression of the effective average loss factor \( \xi P_f \) was determined:

\[
\xi P_f = \frac{\int \Delta p \, dx}{\int \rho \omega^2 \, dx}
\]

For the case under examination the valve loss factor resulted 6.26.

This loss factor is in good agreement with the one coming from the literature and traditionally utilized for the valve loss calculation, which, for this case, was \( \xi P_f = 6.33 \).

![Figure 8. Pressure diagrams resulting from the 3D CFD-FSI analysis](image)

The second way of validation utilized was to compare the calculated value of the compressor power with the value measured in the field.

The compressor brake power \( N_b \) can be calculated as follows:

\[ N_b = N_{ad} + N_v + N_m \]

Where:
- \( N_{ad} \) = adiabatic power
- \( N_v \) = valve + valve port power loss
- \( N_m \) = mechanical power loss

The valve power loss, corresponding to the \( \xi P_f \) previously determined, resulted \( N_v = 88 \) kW. The adiabatic power calculated considering the actual gas characteristics resulted \( N_{ad} = 260 \) kW. The mechanical power loss is calculated in accordance to literature and experience resulted \( N_m = 16.5 \) kW. Thus the compressor brake power resulted \( N_b = 364.5 \) kW which is in good agreement with the measured value of 360 kW.

The new 3D CFD-FSI simulation model is therefore to be considered validated.

### 6. Conclusions

With the help of the CFD-FSI multi-phase analysis, it was possible to demonstrate the hidden dangers of a gas compressor which operates even for a limited number of cycles with ingestion of liquid.

The introduction of approximations utilized to limit the computing time to reasonable values, such as the mix of a Zero-dimensional and 3D CFD analyses or the assumption of the rigid motion of the valve rings has to be considered acceptable for the purpose of this analysis. On the contrary the authors believe that it would be very difficult to treat these elasto-dynamic phenomena with other analytical methods or experimentally.

The authors deem that these models can be effectively extended, providing many benefits, to investigate similar phenomena involving the interaction between multi-phase fluids and flexible mobile components. In particular, the use of this method allows the optimization of fluid dynamic profiles of the valve zone of a compressor cylinder with a good chance of reducing related energy consumption. This new approach can be particularly helpful in the design or for root cause analysis of large compressors handling heavy and/or condensable gases such as the ones utilized in the LDPE, refining and natural gas industries where liquid entrainment can occur for various reasons.

### References