ADVANCES IN CFD ANALYSIS FOR TRANSIENT TURBOCHARGER FLOWS

Fred Mendonça, Stephen Ferguson and Dean Palfreyman
CD adapco Group
London, UK
Contact f.mendonca@uk.cd-adapco.com

1. INTRODUCTION

Transient CFD analysis on turbomachines should be strongly encouraged. It opens up a new chapter of possibilities in the understanding of the workings, efficiency and design optimisation of rotating components.

Turbocharging increases the power output from reciprocating engines by utilizing the waste energy in the exhaust gases. The exhaust gases drive a turbine, connected via a shaft to a compressor, which pressurizes the air at the engine inlet thus allowing more fuel to be burned for the same air/fuel ratio.

Computational analysis on turbocharger turbines and compressors can be performed separately. Engine exhaust flow enters the turbine through a scroll volute connected to the exhaust manifold. The volute discharges high temperature gas, either directly onto the turbine blades or through variable pitch vanes. Interaction between the fixed geometries and wheel creates dynamic flow modes, which are predominantly at the first (single rotation) order or higher (multiple excitations per rotation).

On the compressor side, the decelerating or accelerating vehicle migrates the turbocharger performance towards ‘surge’ or ‘choke’. Under-bonnet space management usually means that the oncoming flow at the compressor face is non-uniform. Slightly eccentric installations or blade damage often results in modal excitation. Both factors lead to a noise problem.

In this article, we explore the benefits of performing transient CFD calculations on both the turbine and compressor sides.

- Improved understanding of the effects of pressure pulses from the engine on the turbine assists in the energy extraction from the exhaust flow.
- Time varying aerodynamic and thermal loading on static and rotating components provides a useful insight into structural excitation.
- Simulation of the acoustical radiation and flow-excited modes inside the engine bay provides a means to help reduce a predominant source of noise.

Enormous value is derived from time-accurate CFD computations, both in their own right and also in the way they are linked with other key design features such as fluid-structure interaction and aeroacoustics. Combined advances in processor speed and turbulence modelling (especially LES-based methods) make it feasible to compute multiple wheel rotations: recent experiences of the present authors using STAR-CD are presented below, and include an example of more than 50 limit-cycled wheel rotations using Detached Eddy Simulation (DES).

In these examples we have employed many Best Practice procedures, including second-order spatial discretisation, y⁺ insensitive near-wall treatments and advanced turbulence modelling.

2. PULSED TURBINES

There is a lack of understanding of the turbine aerodynamics under pulsating conditions. There are two main driving mechanisms, which have different timescales. First, there is the mass exhausted from each cylinder in turn, which convects through the exhaust manifold and volute into the turbine. Secondly, pulsations for the exhaust-valve opening event, propagating at the local speed of sound, are superimposed over the exhaust mass-flow. The period of this pulse is usually an order of magnitude or more faster than the engine period. Therefore the flow and thermal loading on the turbine blades is highly irregular, and also a function of the exhaust and volute geometries. Some designs include guide vanes and volute splitter plates upstream of the blades to even out the oncoming flow.

In a recent study together with Imperial College [1], insight has been gained into the pulsating dynamics of the turbine.

Figure 1: Turbine wheel and volute
The meshed domain encompassing the volute and all turbine passages utilised transient sliding mesh capabilities enabling turbine rotation to be modelled. Figure 1 shows a sample mesh. This work has allowed for the first time an assessment of the propagation of the pulse waveforms through the turbine passages and their interaction with the stationary and rotating components.

The simulation time comprised one complete pulse cycle, during which time the turbocharger completed twelve full rotations. Figure 2 illustrates the passage of a pressure pulse approaching the volute.

A wave-transmissive condition at the domain inlet permits the setting of time-varying velocities from the engine exhaust as well as a measured pulsed pressure upstream of the volute, and allows pressure waves to propagate out through the boundary.

Measurements at Imperial College show that the turbine performance is far from quasi-steady, and in fact exhibits a hysteresis in the efficiency curve during the pulse loading cycle. The effect is very well captured by the transient calculation, as shown in Figure 3. The differences at the lower pressures seen between prediction and measurement are partly due to the fact that the response of the experimental turbine is damped through the shaft connection with the compressor.

For the complete pulse cycle, Figure 4 shows that the predicted tangential velocity at the turbine inlet matches the measurements very well.

### 3. SURFACE EXCITATION

By its very nature, structural excitations by the flow cannot be predicted by any mixing plane or multiple-rotating frames (MRF) of reference approximations, or any other approach which does not employ transient rotating meshes. We present here computations involving a turbine with static vanes and spacers, in which the wakes behind these components interact with the wheel, rotating at nearly 200,000 RPM, causing unsteady blade loading.

Transient computations commence from a representative steady-state solution, usually performed using MRF. Figure 5 demonstrates that a truly limit-cycle solution with unsteady RANS usually takes 5-10 full rotations to evolve.
The unsteady flow data, transformed into the frequency domain, can be used to visualise the spatial blade loading distribution at discrete frequencies. In Figure 6, we see contours of the Power Spectral Density at the frequency corresponding with that at which the blades see the vanes passing.

Figure 6: PSD blade loading

Turbochargers are subjected to narrow-banded and broad-banded flow perturbations. Both may cause modal excitation and aeroacoustics issues (see section 4). As shown previously [2], the choice of turbulence model when applied to unsteady flows can influence the nature of the transient surface excitations in an unphysical way. Unsteady RANS, by overestimating the sub-grid scale viscosity, usually redistributes broad-band spectra into narrow-band features. Large-Eddy Simulation is thus necessary. The differences between the broad-band excitations can be seen clearly by comparing URANS with DES computations for this turbine.

Figure 7: DES Blade pressure over 50 revolutions

Figure 7, shows the DES blade pressure trace for 50 revolutions. The signal is noisy, as one would expect from a LES-based methodology. We continue to follow Best Practices procedures in attempting to ensure that any influence of the initialisation is washed out of the domain. Fourier analyses using windows of 10 rotations indicate that the solution is limit-cycled by about rotation 20, but that it is necessary to execute approximately 30 rotations to extract a suitably large window for the data to be processed smoothly.

For the last 5 rotations, Figure 8 illustrates the differences between unsteady RANS and DES, while in Figure 9, we can see from the Power Spectral Density that the broad-band is captured much better with DES. The latter shows much higher levels of surface excitation across the spectrum.

Calculations from this turbine have been validated insofar as the predicted pressure drop through the turbine is within 2.5% of the measured values.

The model consists of approximately 2.6 million cells. One complete rotation takes approximately 11 hours on 12 processors of a 2.7GHz machine.
4. **ACOUSTICS**

For vans and heavy-duty diesel vehicles, technology advances in injecting systems, particularly common-rail, have dramatically reduced the associated noise emissions. Ironically, this means that turbocharger compressor noise is increasingly becoming an issue.

There are two installation features that can influence the noise emissions from turbocharger compressors. The first comes from eccentric mounting of the wheel, or damage inflicted to blades during installation or operation. Tiny distortions can cause acoustics excitations of a different order from the blade passing frequency (BPF), which then propagate upstream through the intake system and radiate under the bonnet. Secondly, under-hood space management usually determines the flow path from the intake system to the compressor inlet. Non-symmetric on-coming flow loading at the face of the compressor can result in undesirable acoustics emissions.

Furthermore, the vehicle acceleration and deceleration migrates the compressor operating point away from its peak performance towards choking or surge. In the latter case, flow separation on the suction side of the blade will result in acoustics excitations and noise radiation. Aeroacoustic noise source prediction and propagation becomes an issue.

In a recent study of a centrifugal compressor [3], fan noise source generation using STAR-CD was coupled to noise propagation in SYSNOISE to far-field observer locations. Figures 10 and 11 illustrate that good predictions of the sources and propagation can be achieved.

![Figure 10: Propagated noise at the blade passing frequency (BPF)](image)

5. **OUTLOOK**

Transient CFD analysis, by virtue of its practicability, will play an important role in turbomachinery analysis.

![Figure 11: Microphone locations and predicted versus measured noise at the blade passing frequency (BPF)](image)

6. **REFERENCES**

