Large-scale Simulation of Unsteady Flows in 13-stage Multi-spool Compressor

Ziwei Wang\textsuperscript{1,2}, Xiong Jiang\textsuperscript{2*}, Fengshun Lu\textsuperscript{2}, Fan Liu\textsuperscript{2} and Bin Li\textsuperscript{2}

\textsuperscript{1} National University of Defense Technology, Changsha, Hunan, 410073, People’s Republic of China

\textsuperscript{2} China Aerodynamics Research and Development Center, Mianyang, Sichuan, 621000, People’s Republic of China

\textsuperscript{*Corresponding author’s e-mail: jiangxiong@cardc.cn

Abstract. Although significant advances have been made in full-annulus simulation of flow in single-spool compressors. However, no work has been devoted to full-annulus simulation of flow in multi-spool compressors. The aim of the paper is to develop techniques for full-annulus simulation of unsteady flows in multi-spool compressor. The techniques for automatic generation of large-scale full-annulus grid of multi-spool compressor, for uniting simulation in LPC (Low Pressure Compressor) and HPC (High Pressure Compressor) in one solver, for processing LPC/HPC interface in multi-spool compressor in parallel environment, for managing vast amounts of flow field data in multi-spool compressor with low space complexity are proposed. Based on an in-house software ASPAC, these techniques are then applied to full-annulus unsteady simulation of flow in a 13-stage multi-spool compressor with 800 million grid cells using 10240 processes. The numerical simulation results are validated through compared with results of empirical model. Meanwhile, it indicates that results of large-scale unsteady simulation are more precise than results of steady simulation.

1. Introduction
Off-design aero-engine transient simulation is one of the four challenges mentioned in “CFD Vision 2030 Study” proposed by NASA\cite{1}. Lots of flow phenomena in aero-engine under off-design conditions, such as rotating stall and surge in turbomachinery, distortion, transient intersections between components need to be simulated with unsteady methods on full-annulus aero-engine grid.

Significant progress has been made in simulation of flow in aero-engine\cite{2}–\cite{7}. In the early time, different components in aero-engine were simulated by different solvers, intersections between components were realized through exchanging circumferential averaged data. At the beginning, flow in the core engine was simulated with steady methods on single passage grid\cite{2}. After that, the blade number in turbomachinery and number of flame tubes in combustion chamber are scaled to ensure the sector periodicity, then flow in core engine could be calculated with unsteady methods on sector grid\cite{3}. The LPC and LPT (Low Pressure Turbine) were then considered\cite{4}. In the last three years, the simulation of aero-engine were conducted with single solver\cite{5}–\cite{7}, different components were highly coupled during calculation. However, the simulations were still conducted on single passage grid or sector grid. Consequently, the simulation results will deviate from reality, and some full-annulus phenomena in off-design conditions, such as rotating stall and surge, cannot be captured.

As a compromise, the full-annulus unsteady simulation can be implemented on one or more aero-engine components. This paper concentrates on large scale full-annulus unsteady simulation of multi-
stage compressor, some representative studies are listed in table.1, with the study of the paper listed at end of the table. The unsteady full-annulus flow simulation was performed in a 3-stage compressor at the design point with software elsA, the parallel efficiency was tested with up to 4096 processes[8]. Software AU3D was then applied to simulate unstable flow phenomenon in an 8-stage compressor[9].

However, most of the aero-engines contain more than one rotate shaft, meaningly that LPC and HPC should be both taken into consideration during simulation. The configurations of single-spool compressor and multi-spool compressor are shown in Figure 1.

| Year  | LPC          | HPC          | grid number | number of processes |
|-------|--------------|--------------|-------------|--------------------|
| 2010  | not include  | 3-stage      | 134 million | 4096               |
| 2020  | not included | 8-stage      | 300 million | 192 Intel Xeon CPUs|
| current | 4-stage     | 9-stage      | 800 million | 10240              |

To the best knowledge of authors, no work has been devoted to full-annulus unsteady simulation of multi-spool compressor. Which can be used in future study to investigate full-annulus phenomena under off-design conditions in multi-spool compressors that cannot be modelled with single passage grid or sector grid.

Compared with single-spool compressors, full-annulus simulation of flow in multi-spool compressors will bring some difficulties, listed as follows:

- the rotation speeds and configurations of LPC and HPC are different, making it hard to unite simulation procedures in one flow solver
- real time and accurate data exchange between LPC and HPC
- more compressor stages means that more grid number and more processes are needed in simulation, bringing difficulties about grid generation, large-scale simulation and post-process of vast amounts of flow field data.

To solve the problems mentioned above, a load-balanced grid decomposition algorithm considering compressor configuration characters is developed, a technique to unite simulation of LPC and HPC in one solver is proposed, a parallel strategy for large-scale simulation and a numerical algorithm for real time and accurate data exchange between LPC and HPC are developed, and an algorithm for processing vast amounts of flow field data is developed.

The Paper is organized as follows. In section 2, the numerical methodology is described. Section 3 presents the full-annulus simulation of a 13-stage multi-spool compressor, and shows a comparison with the standard industrial approach (a steady flow simulation method: mixing plane method). Some conclusions are drawn in Sec. 4.

2. Numerical methodology for large-scale simulation of flow in multi-spool compressor

2.1. Flow solver description
The simulation is conducted based on the in-house software ASPAC[10], which had been validated for various turbomachinery flows. The 3-D compressible Navier-Stokes equations is solved by fully-implicit scheme with a cell-centered finite volume method. The parallel is conducted based on MPI.
2.2. Meshing strategy

2.2.1. Grid generation

The structured multiblock grid is used for modelling the complex configuration of multi-spool compressor. NUMECA AutoGrid is chosen to generate grid. However, NUMECA is usually installed on PC, the data amount of full-annulus multi-spool compressor grid may exceed memory of PC. To solve the problem, the single passage grid is generated with NUMECA AutoGrid. The full-annulus grid is then generated on workstation, the procedures are shown in Figure 2.

![Figure 2 Procedures for generating full-annulus grid.](image)

2.2.2. Grid decomposition

The parallel algorithm is conducted based domain decomposition method. After grid decomposition, grid cells in different domain are calculated with different process.

In terms of structured multiblock grid, the load balance is obtained by greedy algorithm through splitting the biggest blocks at every step. Some adjustments are introduced through taking configuration of multi-spool compressor into consideration. Blocks belong to different row interfaces will not be placed in the same process. When number of processes is relatively large (>2048), the degree of load balance will hardly be influenced.

It needs to be mentioned that, in some former studies [11], to make data exchange at row interface easier, the block at row interface was directly generated with 360° sector and couldn’t be split, thus the degree of load balance will be destroyed. Which can be avoided in the algorithm mentioned above.

2.3. Large-scale simulation strategy

As shown in Figure 1, to manage the complex configuration of multi-spool compressor in single flow solver, the idea of object oriented is introduced. The blocks are categorized to different row domains, each row domain is taken as an object with its own properties, such as rotation speed, blade number and so on. Moreover, though grid in one process may contain blocks from different blade row, each row domain is solved separately.

Row interfaces are used to manipulate data exchange between blade row domains. Each row domain is taken as an object with its own properties. The difference between row interface in single-spool compressor and multi-spool compressor is shown in Figure 3. In single-spool compressor, the blocks adjacent to row interface have a rigid rotation speed or don’t rotate. However, at LPC/HPC interface, the blocks adjacent to row interface have more rotation speeds at different radial regions, and flow data needs to be exchanged at every time step between upstream and downstream blocks adjacent to interface. An algorithm is proposed to manage data exchange at LPC/HPC interface.

- the blocks are generated restricted by the rule that block domain don’t intersect with more than one rotate region
- the interface is split along the dividing line of speed in radial direction
- the split interfaces are managed separately
- taking blocks at one side of row interface as stationary, calculate the relative movement of blocks at opposite side at every time step, then calculate data exchange
The parallel calculation of row domains and row interfaces are uncoupled to avoid global communication and communication conflicts, as shown in Figure 4. Different row interfaces are calculated at different process groups to decrease space complexity of dynamic data interpolation between blocks adjacent to row interfaces. The non-blocking communication and data packaging technology are developed. The parallel techniques developed is proved to be successfully used for stable operation of simulating multi-spool compressor with ten thousand processes.

2.4. Strategy for processing vast amounts of flow field data
As proposed in “CFD Vision 2030 Study”[1], the methods to manage vast amounts of data generated by large-scale simulation are lacked.

To solve the problem in multi-spool compressor, the row domains are processed separately. However, the data in one row domain may be distributed in different processes and one process may contain data from different row domains. The pseudo-FORTRAN 90 code for solving the problem mentioned is shown as follows:

```fortran
    do nProc = 1,num_processes !Iteration of process
        call GatherFieldAndGridData(nProc) !Read flow and grid data of current process from corresponding files
        call ProcessEachBlock(nProc) !Process grid and field data of blocks, then output to corresponding row directory
        call FreeProcData(nProc) !Free memory allocated
    end do

call ReadBlockConnectivityData !Input block connectivity information of all blocks, memory needed is O(10) MB

do nRow = 1,num_row !Iteration of all the row domains
do nb=1,num_blocks(nRow) !Iteration of blocks belong to domain of current row
    call ProcessFieldAndGridData(nb) !Read data of current block and process it based on block connectivity
    call ExtractAnalysisInformation(nb) !Extract information needed for analysis and output
    call FreeProcData(nb) !Free memory allocated
end do
end do

call CombineAnalysisInformation !Combine and process analysis information of all blocks, then output it to files
```

Based on the algorithm mentioned above, the time complexity increases, however the space complexity is largely reduced to $1 / \min(\text{num\_processes}, \text{num\_blocks})$ of the original.

3. Application to unsteady flow simulation of multi-spool compressor
The flow in multi-spool compressor at design-condition is simulated with full-annulus method, the results is validated through compared with results of 2D empirical model and steady simulation.

3.1. Test case
The multi-spool compressor is designed by CARDC(China Aerodynamics Research and Development Center), which consists of 4-stage LPC and 9-stage HPC, the design speed is 9500rpm for LPC and -19078rpm for HPC, the design pressure ratio is 3.55 for LPC and 15.5 for HPC.
3.2. Simulation results

The single passage grid is firstly generated by NUMECA AutoGrid, as shown in Figure 5. The total grid number is 24 million, which is used for steady simulation. The full-annulus grid is then generated based on ASPAC with the same boundary conditions. The inlet total pressure is 101325Pa, the inlet total is 288.15K, the static pressure at hub of external duct outlet is 33000Pa, the static pressure at hub of internal duct outlet is 5500800Pa. The Spalart-Allmaras turbulence model is applied.

At rotor/stator interface, the variables are circumferential averaged to become 1D radially distributed and then exchanged in steady simulation, the variables are dynamically exchanged through interpolation at every time step in full-annulus simulation.

The design parameters are generated through 2D empirical model, which is not as precise as experiments, but is physical and can be used to validate the simulation results.

Comparisons among 2D empirical model, steady simulation and full-annulus unsteady simulation are listed as Table 2. The parameters of steady simulation and full-annulus unsteady simulation match well with 2D empirical model, indicating that the simulation results are physical.

The difference between mass flow in inlet and outlet are -1.92% for steady simulation and 0.15% for full-annulus unsteady simulation. The full annulus method can get more precise results.

Table 2 Comparisons among different calculation results

|                         | empirical model | Steady simulation | Unsteady simulation |
|-------------------------|-----------------|-------------------|--------------------|
| Mass flow at inlet      | 120kg/s         | 121.28kg/s        | 121.28kg/s         |
| Mass flow at LPC outlet | 40kg/s          | 36.61kg/s         | 36.54kg/s          |
| Mass flow at HPC outlet | 80kg/s          | 87.00kg/s         | 84.56kg/s          |
| Pressure ratio of LPC   | 3.55            | 3.53              | 3.53               |
| Pressure ratio of HPC   | 15.5            | 15.42             | 15.43              |

The corresponding total size of grid files is about 18GB, total size of flow field files is about 30GB, number of blocks is about 18000 before grid decomposition. To post process the flow field, the complexity for store flow data and grid data is about O((18GB+30GB)/min(10240,18000))=O(4.8MB), which can be easily realized on PC.

The combined analysis results are listed in Figure 6, which is the entropy distribution at half span calculated by both methods. For steady simulation, the blade wakes are cut off at rotor/stator interface, which is different from reality and computing error accumulates during the process. For full-annulus unsteady simulation, the blade intersections are clearly captured, moreover, the interactions between LPC and HPC are also correctly captured.
4. Conclusions
In this paper, based on the in-house software ASPAC, a series of techniques for simulating multi-spool compressor are proposed, which are then applied to full-annulus unsteady simulation of flow in 13-stage multi-spool compressor.

In terms of simulation of the 13-stage multi-spool compressor, different from former study[11], the degree of load balance is ensured without influencing data interface manipulation. The parallel techniques developed is proved to be successfully used in stable operation of simulation on 10240 processes. the space complexity for post-process of flow field data and grid data is decreased to about O(4.8MB).

Through compared with the results of 2D empirical model, the numerical simulation results are validated. Meanwhile, it is proved that the results of large-scale full-annulus unsteady simulation are more precise than those of industrial steady simulation. In the future, these techniques will be applied to analysis of flow phenomena in multi-spool compressor at off-design condition and off-design aero-engine transient simulation.

Acknowledgments
Authors wishing to acknowledge enlightening discussion from Dr. Nianhua Wang.

References
[1] Slotnick, J., Khodadoust, A., Alonso, J., et al., (2014) CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences.
http://www.nasa.gov/aero/research/programs/tacp/td/cfd-vision-2030-study/
[2] Turner, M., Reed, J.A., Ryder, R., et al., (2004) Multi-Fidelity Simulation of a Turbofan Engine With Results Zoomed Into Mini-Maps for a Zero-D Cycle Simulation, NASA/TM—2004-213076, GT2004–53956.
[3] Schluter, J., Wu, X.H., Weide, E., et al., (2006) Integrated LES-RANS of an entire high-spool of a gas turbine. In: 44th AIAA aerospace sciences meeting and exhibit, Reno.
[4] Medic, G., Kalitzin, G., You, D., et al., (2007) Integrated RANS/LES computations of an entire gas turbine jet engine. In: 45th AIAA aerospace sciences meeting and exhibit, Reno
[5] Orlov, M.Y., Lukachey, S.V., Anisimov, V.M., (2018) Investigation of the effect of the non-uniform flow distribution after compressor of gas turbine engine on inlet parameters of the turbine. In: IOP conference Series: materials science and engineering. Bristol.
[6] Teixeira, M., Romagnosi, L., Mezine, M.A., (2018) A methodology for fully-coupled CFD engine simulations, applied to a micro gas turbine engine. In: ASME Turbo Expo: Turbomachinery technical conference and exposition. New York.
[7] Perez, C.A., Dombard, J., Duchaine, F., et al., (2020) Large-eddy simulation of an integrated high-pressure compressor and combustion chamber of a typical turbine engine architecture. In: ASME Turbo Expo GT2020-16288. London.
[8] Gourdain, N., Montagnac, M., Gzaix, M., (2010) High-performance Computing to Simulate Large-scale Industrial Flows in Multistage Compressors. INT J HIGH PERFORM C., 24(4): 429-443.
[9] Zhao, F., Dodds, J., Vahdati, M., (2020) Flow Physics During Surge and Recovery of a Multi-Stage High-Speed Compressor. In: ASME Turbo Expo 2020: Turbomachinery Technical Conference and Exposition. Online.
[10] Wang, Z., Fan, Z., Jiang, X., (2017) Investigation on harmonic balance method for simulation of unsteady flow in the multi-stage compressor. Journal of Propulsion Technology., 38(11): 2512-2521.
[11] Espinal, D., Im, H. S., (2018) Full-Annulus Simulation of Nonsynchronous Blade Vibration Excitation of an Axial Compressor. J TURBOMACH., 140(3):031008.