1. **Participant Information**

|  |  |
| --- | --- |
| First name: | Boyang |
| Last name: | Liu |
| Organization: | Rankyee Technology, LTD. |
| Email: | liuboyang@rankyee.com |
| Submission year: | 2024 |

1. **Grid Information**

If this submission used an official grid:

|  |  |
| --- | --- |
| Grid name (e.g., medium, fine, etc.): | Ultracoarse |
| Has pinched rotor casing?\* (yes/no) | yes |
| Has stator hub cavity? (yes/no) | no |

\* The official grids released in the 2021 1st GPPS CFD Workshop used a smooth rotor casing (not realistic). The official grids released in the 2022 2nd GPPS CFD Workshop has fixed this error with realistic pinched casing.

If this submission used an in-house grid:

|  |  |
| --- | --- |
| Average y+ of the first layer grid: | About 30 based on the degree of freedom |
| Number of grid points in the rotor domain: | 1,200,000 elements, 76,800,000 degrees of freedom |
| Number of grid points in the stator domain: | 1,000,000 elements, 64,000,000 degrees of freedom |
| Type of grid element:  (e.g., hexahedron, tetrahedron, etc.) | hexahedron |
| Has pinched rotor casing?\* (yes/no) | yes |
| Has realistic rotor and stator fillets? (yes/no) | Add fillets for rotor |

1. **Flow Solver Information**

(1) General:

|  |  |
| --- | --- |
| Solver name: | Dimaxer |
| Version number: | 2024R1 |
| Major reference(s) (optional): | Y. Lu, K. Liu., 2020, “KEPLER: An integrated simulating system for accurate and efficient industrial design and optimization”, AIAA-SciTech-Paper. |

(2) Advection Scheme:

|  |  |
| --- | --- |
| Branch of scheme (e.g., JST, ROE, AUSM): | HLLC |
| If not listed above, please briefly describe the advection scheme and include a major reference to the scheme: | |

(3) Turbulence Model:

|  |  |
| --- | --- |
| Type of scale-resolving simulation (e.g., SAS. SBES, DDES, LES, etc.): | LES |
| RANS branch model name (e.g., SA-noft2, SST-2003)\*, if applicable: | None |
| If not documented in NASA TMR, please briefly describe the RANS turbulence model and include a major reference to it: | |
| LES branch model name (e.g., implicit, Smagorinsky, WALE, etc.): | WALE |
| Please briefly describe the LES SGS turbulence model and include a major reference to it: | |
| RANS-LES interface model (e.g., shielding functions, acceleration functions), if applicable: | None |
| Please briefly describe the LES SGS turbulence model and include a major reference to it: | |

\*Please follow the naming convention of [NASA TMR](https://turbmodels.larc.nasa.gov/). Note that the turbulence model implemented in the solver may differ from the standard version of the model (e.g., SA vs. SA-noft2, SST vs. SST-2003, etc.)

(4) Viscous wall treatment:

|  |  |
| --- | --- |
| Use of wall function (yes/no): | yes |
| Use in-house grid with y+ > 10 (yes/no): | yes |
| If both yes, please briefly describe the wall function and include a major reference to it:  [Wall-stress model: ODE – WMLES (umd.edu)](https://wmles.umd.edu/wall-stress-models/wall-model-ode/) | |

(5) Rotor-stator interface model:

|  |  |
| --- | --- |
| Type of model for mean flow quantities\*:  (e.g., frozen rotor, mixing plane, non-reflecting (Giles)) | Dynamic Overset |
| Type of model for turbulence quantities\*:  (e.g., frozen rotor, mixing plane) | Dynamic Overset |
| Please briefly describe the rotor-stator interface model and include a major reference to it (optional): | |

\* Mean flow quantities are *p*, *T*, *u*x, *u*y, *u*z, etc.; turbulence quantities are eddy viscosity, *k*, *ω*, etc.

(6) Other details (optional):

|  |  |
| --- | --- |
| Fluid model (e.g., real gas, idea gas): | Ideal gas |
| Linear system solver (e.g., Jacobi, etc.): | None |
| Have you verified your solver in [NASA 2D flat plate](https://turbmodels.larc.nasa.gov/flatplate.html) against established RANS solvers? (yes/no) | yes |

(7) Temporal Discretization Scheme:

|  |  |
| --- | --- |
| Branch of scheme (e.g., Runge-Kutta, Backward Euler, etc.): | Runge-Kutta |
| If not listed above, please briefly describe the advection scheme and include a major reference to the scheme: | |
| Order of accuracy: | Fourth |
| Time step used in simulations (in second): | Local time step |

1. **Boundary conditions**

(1) Inlet:

|  |  |
| --- | --- |
| How were the mean flow quantities determined? (e.g., from InletBC.input file; uniform inlet at standard conditions) | Uniform inlet at standard conditions |
| How was the turbulence quantity(s) determined? (e.g., values and units of inlet *k* and *ω*) | Laminar inlet |

(2) Outlet (optional):

|  |  |
| --- | --- |
| What type of boundary condition is used? (e.g., uniform backpressure, radial equilibrium backpressure, mass flow, Riemann, etc.) | uniform backpressure |

(3) Periodic boundary (optional):

|  |  |
| --- | --- |
| Have you checked the periodicity of mean flow quantities? (yes/no) | no |
| Have you checked the periodicity of turbulence quantities? (yes/no) | No |

1. **Convergence History**

A figure of mass flow rate (rotor inlet, rotor exit/stator inlet, and stator exit) versus Time

|  |
| --- |
| Peak efficiency condition (16.00 ± 0.10 kg/s) |
|  |

A figure of residual versus iteration (optional)

|  |  |
| --- | --- |
| Which quantity's residuals are plotted? |  |
| How was the residual defined? (global or local; maximum or average; absolute or relative, etc.) |  |
| Peak efficiency condition (16.00 ± 0.10 kg/s) | Near stall condition (14.78 ± 0.10 kg/s) |
|  |  |