Instructions for MISES Analysis of Case 1.2.TT

1 Prerequisites

1. T-Blade3 executables (tblade3, techop) can be installed following the instructions in the documentation available at `gtsl.ase.uc.edu/t-blade3/T-Blade3_v1.2_ReadMe.pdf`.
2. MISES is not open-source. Licensing information is available at `https://tlo.mit.edu/technologies/mises-software-design-and-analysis-turbomachinery-blading`. The executables iset, ises, iprint, iplot are required. The steps outlined were carried out in MISES v2.57.

2 Geometry Generation

1. Run the following commands based on the location of the T-Blade3 executables:
   $ /path/to/executables/tblade3 3dbgbinput.2.dat
   $ /path/to/executables/techop blade.1.2.TT 15

3 MISES Analysis

This is a summary of the steps required to run MISES for this particular case. Further information about all the executables and files can be found in the documentation at `web.mit.edu/drela/Public/web/mises/mises.pdf`.

1. Place the files blade.1.2.TT and ises.1.2.TT in a separate directory.
2. Run iset first to generate the grid and an initial idat.1.2.TT file. Run the following command:
   $ /path/to/executables/iset 1.2.TT
   This will open a XWindow instance showing the cascade and open a menu in iset. Next, in the iset menu:
   (a) Select option *2 Generate spacings and initialize grid*. Hit enter without making any changes to the spacings.
   (b) Select option *3 Elliptic grid smoother*.
   (c) Select option *4 Write idat.1.2.TT*.
   (d) Select option *5 Plot grid*. This plots the generated grid in the previously opened XWindow instance and opens a sub-menu in iset. Hit enter to quit the sub-menu without making any changes.
   (e) Select option *8 Write grid parameters to gridpar.1.2.TT*.
   (f) Select option *0* to quit iset.
3. This has created two files: gridpar.1.2.TT and idat.1.2.TT.
4. Next, launch ises by running the following command:
   $ /path/to/executables/ises 1.2.TT
   The boundary conditions for this case are:
   (a) Drive inlet slope to the inlet slope value specified in ises.1.2.TT ($S1$).
   (b) Use TE Kutta condition for the current blade.
   (c) Use LE Kutta condition for the current blade.
   (d) Drive exit pressure ratio to value specified in ises.1.2.TT ($P2/Po1$).

   ises requires the number of iterations as input. The recommended initial iteration count is 50. For higher positive and negative incidences, more iterations might be required.
5. Once ises has converged, run iprint to print a summary of the forces on the blade including the loss coefficients $\bar{\omega}$ and ζ. This summary can be saved in a file for future reference on Linux/Unix systems by running the following command:
   $ /path/to/executables/iprint 1.2.TT > output_filename

6. Finally, run iplot to generate blade surface and contour plots of quantities like Mach number, C_p or C_f. Run:
   $ /path/to/executables/iplot 1.2.TT
   This opens a menu in iplot. To generate a blade surface Mach number plot:

   (a) Select option *1 Blade surface plots*. This opens a sub-menu.

   (b) In the sub-menu, select option *1 Mach vs x*. This will plot the Mach number distribution in a XWindow instance.

   (c) To save the current plot in a postscript file plot.ps, select option *14 Hardcopy current plot*.

   (d) Select the appropriate option to plot another quantity or select option *0* to quit to the top-level iplot menu.

   To generate a contour Mach number plot:

   (a) Select option *3 Contour/grid plots* in the top-level iplot menu. This opens a sub-menu.

   (b) In the sub-menu, select option *3 Flow contours (MPRQCLT)*. This opens another sub-menu.

   (c) In the contour/grid plot option sub-menu, select option *M* to plot the cascade Mach number contour in a XWindow instance.

   (d) To go back to the contour/grid plot option list, select *-1* or select *0* to go back to the top-level iplot menu.

   To quit iplot, select *0* in the top-level iplot menu.