AutoGrid Mesh Generation

Autogrid software was used to mesh the turbomachinery section of the ducted fan.

1. **Making the mesh**
    - Import the geometry curves into Autogrid and assign to correct parts
        - Right click row 1 > import & link CAD
        - Load geometry and link hub, shroud, blade, LE and TE
        - Right click row 1 > properties then define properties
        - Properties allows you to edit streamwise number of block points
        - For inlet and outlet the z curves do not need to be fixed geometry

    - Generate B2B
        - Make sure FNMB connection is turned off
    - Generate 3D
    - Check mesh quality
        - Mesh quality can only be checked once 3D is generated
        - Click the small blue button in the left side of the toolbar which looks like a bar chart, a scale will appear on the 3D viewer and a window should open
        - Can change to check orthogonality, aspect ratio etc.
        - Values should be in the following range:
            Skewness >= 20
            Aspect ratio = approx. 500
            Expansion < 2.5

2. **Converting files to correct format**

    - Once mesh is produced, the file must be exported to IGG and in the correct format
    - Most solvers are in right hand coordinate system but TURBOSTREAM works in left hand. Autogrid blocks must be manually converted into left hand only
        - Modules > IGG > Open file > save file again within IGG
        - File > Export > PLOT3D > click folder icon next to PLOT3D File and save as file_name.g in selected location
        - The above creates two files in the chosen location 'file_name.g' and 'file_name.bcs'

    - TURBOSTREAM input files must be '.xdmf' and '.hdf5', therefore the IGG files '.g' and '.bcs' must be converted to suitable TURBOSTREAM input files. This can be done using a python script called 'ag_convert_ts.py'
        - Copy IGG files from where they are saved (home computer) to Darwin (remote super computer) using local terminal (scp)
        - Log into Darwin terminal from local (ssh) and go into eVTOL folder, where the '.g' and '.bcs' files should be (cd)
        - Load TurboStream environment

source /rds/project/hpc/rds-hpc-pullan/ts3/bashrc_module_ts362

- Within TurboStream environment (ts3), use python script to convert '.g' and '.bcs' to correct format for TurboStream, .xdmf and .hdf5

python ag_convert_ts.py file_name

- Copy these files from Darwin to home computer (scp)

3. From MATLAB, run .xdmf and .hdf5 on TURBOSTREAM

- 'Run_Datum' is used to apply boundary conditions and run the mesh on TURBOSTREAM