User manual: QSQH synthetic model

Yunjiu Yang

December 12, 2023

1 Introduction

1.1 Purpose

QSQH synthetic model is a synthetic turbulence model, allowing you to generate wall-bounded turbulent velocity field for any Reynolds numbers. This user manual provides instructions on how to use the software of this model.

1.2 System Requirements

- Environment: MATLAB 2023b or later
- RAM: 16 GB or more
- Disk Space: 10GB or more

1.3 Installation

1. Download the main program from the latest release available at GitHub.
2. Download the test input also from Google drive.
3. Download the 1D universal velocity field from Google drive.
4. Download the 3D universal velocity field from Google drive.

2 Getting Started

2.1 Launching the code

To launch the code, launch MATLAB on you own device first. Open the "`Interface.m`" file in MATLAB.

2.2 User Interface Overview

The interface consists of two sections:

- Data converter: this section will convert the input data to the data suitable for the QSQH synthetic model.
- QSQH synthetic model: this section will generate the instances of the synthetic turbulent velocity field.

2.3 Testing the software

You can use the testing dataset as the input to the software, and test the functions of it. Familiarize yourself with the layout to navigate the software effectively. You are required to enter the directories of the input and output in the designated blank spaces on the interface, no specific file names are required unless otherwise noted. Follow the steps:

1. Enter the directory of the testing input data at "`dir_input`" .
2. Enter the directory for storing the converted input data at "`dir_save`" .
3. Enter the stream-wise and span-wise domain size of the field of the input data at "`Lx`" and "`Lz`" respectively, in wall unit.
4. Enter the directory of the universal velocity field at "`dir_vel_tilde`" .
5. Enter the directory of the u_{τ_L} and θ at "`dir_u_tauL`" .
6. Enter the directory of the statistics of u_{τ_L} and θ at "`dir_u_tauL_stats`" .
7. Enter the directory for storing the testing output data at "`dir_output`" .
8. Enter the maximum stream-wise, wall-normal and span-wise positions of the synthetic velocity field in wall unit, at "`xmax`", "`ymax`" and "`zmax`" respectively. The values of `xmax`

and `zmax` should be set to 0 for generating the 1D QSQH synthetic velocity field. If 3D QSQH synthetic velocity field is to be generated, `xmax` and `zmax` are preferrable to be set the same as the spatial size of the 3D universal velocity field, that $x^+_{\max} = 204$ and $z^+_{\max} = 120$. The value of "`ymax`" should not exceed 100 in wall unit.

9. Run "`Interface.m`".

2.4 Output

The output of the 1D QSQH synthetic model will be the instances of the synthetic velocity profiles in the wall-normal direction, where the outermost position $y^+_{\max}$ is defined by the user. The output matrix files are of the size: $N_t \times N_\tau \times N_y$, where N_t is the number of snapshots of the input wall friction ($N_t = 1$ for the test input), and N_τ is the number of points from which the wall friction is sampled, therefore, $N_t \times N_\tau$ yields the number of instances generated by the model. Note that N_y is the grid points of the synthetic velocity profiles in wall-normal direction.

The output of the 3D QSQH synthetic model will be the instances of the synthetic velocity field of a uniform grid of the size: $x^+_{\max} \times z^+_{\max} \times y^+_{\max}$, with $\Delta x^+ = 12$ and $\Delta z^+ = 6$. The output matrix files are of the size: $N_t \times N_\tau \times N_x \times N_z \times N_y$, where $N_x \times N_z \times N_y$ are the grid points of the synthetic velocity field in x, z and y directions.

The QSQH model includes the re-scaling of the velocity field by the corresponding large-scale wall friction, meaning the instances of the synthetic field will stretch or shrink uniformly by the same factor in x, y and z as comparing to the universal velocity field, therefore, they are interpolated to a uniform grid as mentioned in Step 4. The size of the uniform grid is generally larger than the original grids of the instances of the synthetic field in x and z direction, hence there are "`NaN`" at (x^+, z^+) positions in the output instances, which should be neglected in the calculation of statistics.

2.5 Preparing your own input data

You should prepare the data of wall shear stresses in x and z directions, as different snapshots in the form of MATLAB matrix files of the size of $N^*_x \times N^*_z$, where N^*_x and N^*_z are the number of grid points in stream-wise and span-wise directions of the wall friction field. The matrix files should be named as: "`tau_xxxxx.mat`" where "`xxxxx`" stands for the time or the indices of the snapshots. Note that the input data should be prepared in wall units. Once the data is prepared, you can replace the testing input data with your own input data in step 1, and repeat step 2 to 7 to generate the synthetic turbulent velocity field corresponding to the input wall friction.