Joseph Leonor
ME_395
HW2 – Propose a real-life problem

Characterizing fluid flow has been actively researched for a long time. The study of fluid dynamics has branched off into many specialties that serve other fields such as aerospace and manufacturing. Since I am expecting to research within the field of fluid dynamics, I wanted to use computational fluid dynamics (CFD) software to collect data. The problem I propose is not what I plan to pursue for my PhD research, but it is a representative problem that computational fluid models try to characterize. Out of the classic problems, one which I intend to investigate with the six modules of mechanistic data science is characterizing fluid flow around a cylinder. Characterizing fluid flow around objects is essential for many modern forms of transportation (such as airplanes). Using a cylinder will allow me to better verify the results theoretically.

These CFD simulations output a lot of data. Examples of outputs include velocity in the x- and y-directions, dynamic viscosity, shear velocity, and pressure at each mesh node. Moreover, we have inputs such as fluid density, the geometry of the experiment, and boundary conditions. As a result, it is possible to classify the fluid flow as laminar, turbulent, or somewhere in between with these outputs and to provide an estimate of the Reynolds number for the entire system. Without supplying the theoretical Reynolds number (which is calculated as the product of the velocity and diameter divided by the kinematic viscosity) as an input, it is possible to determine if these classifications match our expectations.

With all these data points, we should satisfy module 1: multimodal data generation and collection, and module 2: extraction of mechanistic features. To fulfill module 3: knowledge-driven dimensional reduction, we can narrow down the number of expected features we will use in our regression. This can happen by determining relationships between the theoretical Reynolds number and other variables in our database. Moreover, it's possible to determine which nodes in the mesh will be particularly indicative of the type of fluid flow.

I then plan to complete modules 4 and 5, which concern creating regression models using neural networks and reduced order models, using parameters I identified to be related to the desired outcome. From these modules, I plan to have outputs classifying the input fluid flow and predicting the associated Reynolds number. These outputs can later be used in module 6: system and design, to inform decision making regarding this CFD model. Moreover, it may be able to inform actual experiments (which is typically the goal of CFD simulations). For instance, if a fluid were to have markers representing particles in an actual experiment, computer vision could possibly perform a task similar to what I am now performing on my simulated model. We would ideally be able to determine the directional velocity at each "node," and apply a similar approach to generalize the model. Hence, investigating this problem can have real impact outside the simulated realm, and afterwards, subsequent models can build on the work already completed.

References:
https://en.wikipedia.org/wiki/Reynolds_number#In_viscous_fluids (article, see stirred vessel)
https://en.wikipedia.org/wiki/Turbulence_modeling (for better understanding of the turbulence model used – Prandtl's mixing-length concept)
https://www.flow3d.com/ (licensed software used to generate data, fluid database)
https://www.smooth-on.com/page/viscosity-scale/ (viscosity for molasses)
https://physics.info/density/ (densities, molasses taken for sugar)