Using Convolutional Neural Network to Predict Unknown Upstream Events

Reza Nouri and Ahmad Vasel-Be-Hagh (Faculty Advisor), Fluid Mechanics Research Lab, Mechanical Engineering Department, Tennessee Tech University

Background

Artificial Intelligence is a promising way to investigate the underlying features behind an occurring phenomenon that seems unpredictable to other ordinary tools, especially in fluid mechanics. Deducing information about upstream events is an ongoing challenge. The direct observation of some upstream events could be impossible or extremely dangerous. For example, if the environment in which the upstream event happens is toxic or radioactive. Convolutional Neural Network (CNN) is a class of deep learning AI trained and used in this study to predict the unknown geometry in an upstream event in a fluid flow.

What is the Problem?

This research has two focus areas:

(i) Application of an existing CNN tool, namely GoogLeNet, on a 2D flow where square and circle bluff bodies were placed upstream (in two different cases) to generate a chaotic flow

(ii) Using GoogLeNet in a fully turbulent and utility-scale wind farm to predict the yaw misalignment of wind turbines within the farm. This research aims at obtaining information about the upstream events by analyzing fluid velocity downstream.

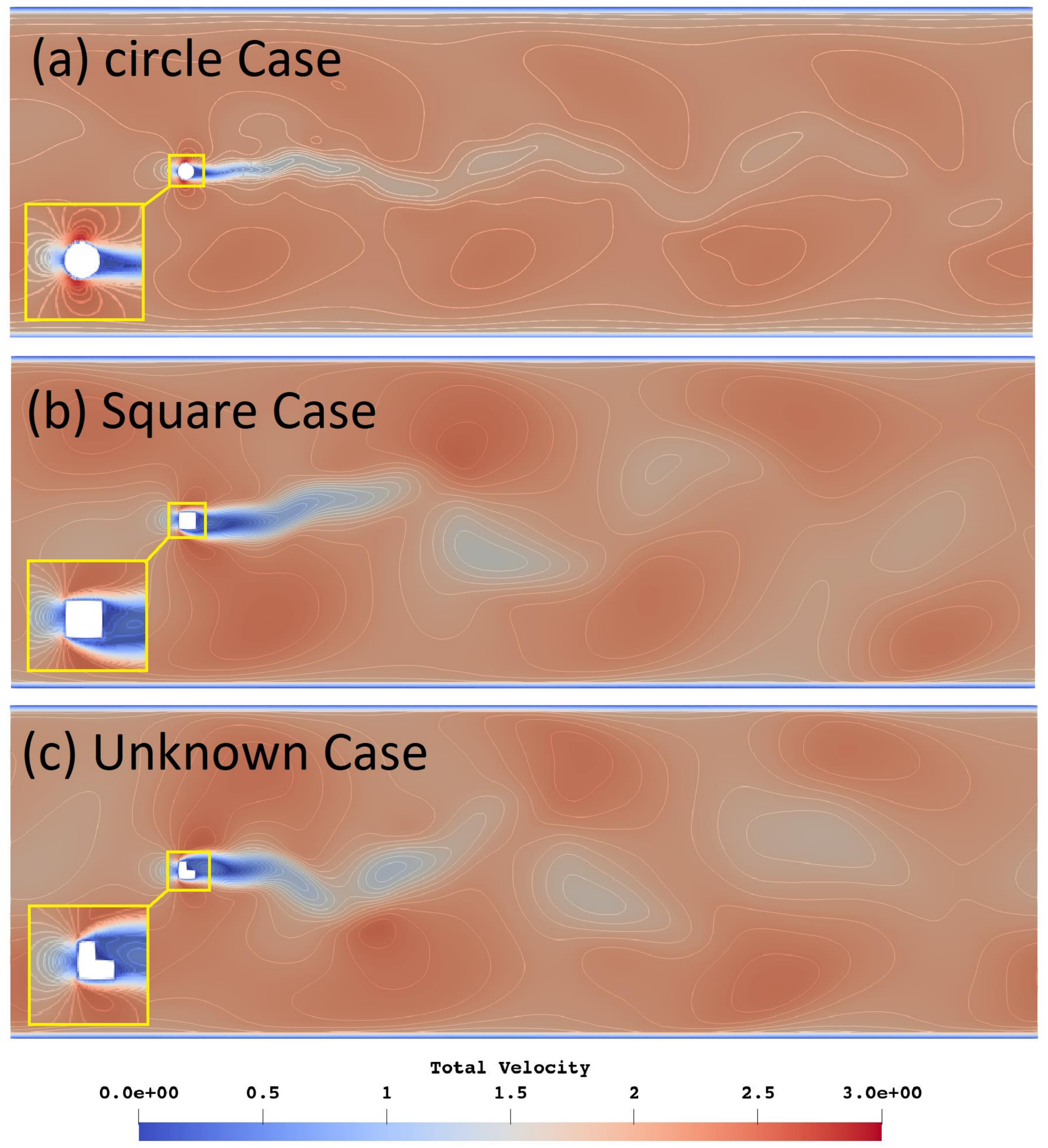

Figure 1. Two cases used to train the model and the unknown case to test the model

How to apply GooLeNet?

Velocity signals were recorded for 30 seconds at 10,000 HZ frequency downstream the flow close to the outlet. A total number of 56 signals (28 signals from the circle case and 28 from the square case) were used to train the model. The input of GoogLeNet is the absolute value of the continuous wavelet transform (CWT) of velocity signals. CWT transforms velocity signals into functions of frequency and time called scalograms. After training the model, the velocity signals of the unknown case (in the eye of the model) were fed to the model, and GoogLeNet predicted the similarity of the unknown shape to both circle and square geometries

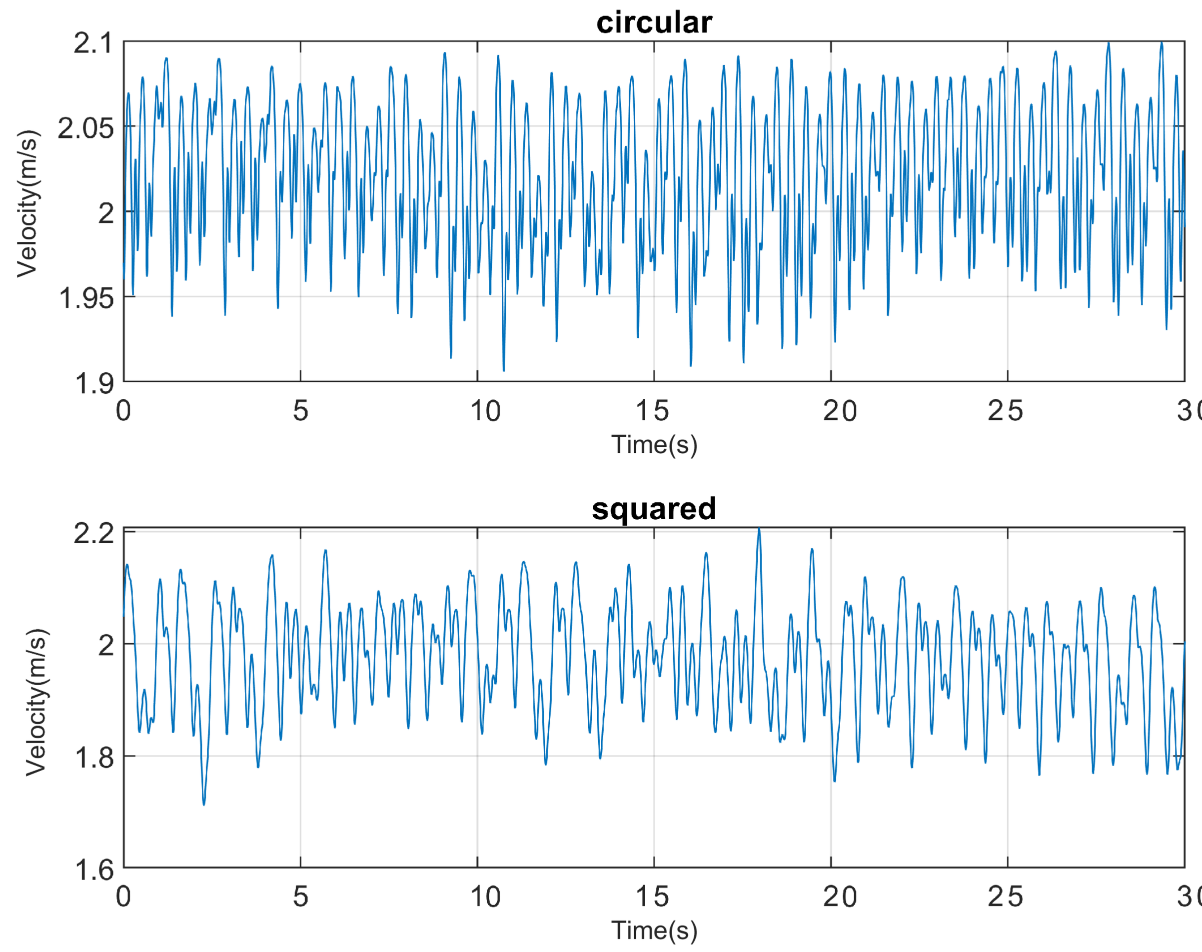

Figure 2. A velocity signal for the circle case (top) and square case (bottom)

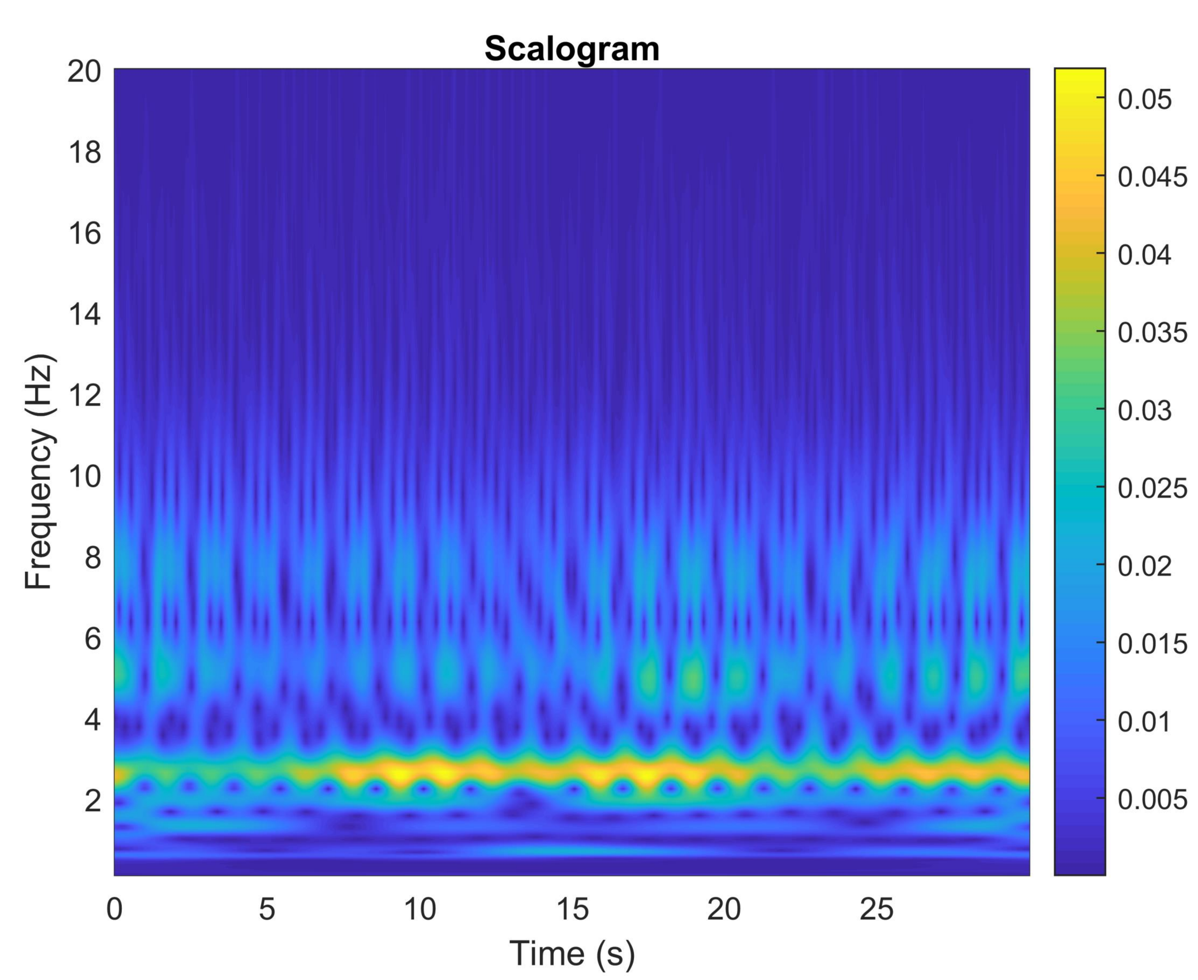

Figure 3. An scalogram generated for a random signal from circle case

Methods

Cases Three CFD simulations were conducted using OpenFOAM (an open-source C++ package) to record the velocity signals.

Domain A rectangle with length and width of 305 cm and 99 cm were selected as the computational domain. The diameter of the circle and side of the square were both 10 cm.

Mesh The mesh for each case was generated by the blockMesh utility provided by OpenFOAM. The areas near the bluff bodies (e.g., square and circle) were refined to ensure that the grid has enough resolution. This yielded a total of approximately 40,000 cells for each case.

Simulation Each case ran for 30 seconds in real-time, and 28 velocity signals of each case were recorded.

Some Selected Results

Table 1 shows some of the predictions of the model for unknown signals. As expected, the unknown geometry (a square missing a corner, see Fig. 2) was predicted to be more similar to a square than a circle, which was an outstanding prediction and 100% correct for all the unknown signals tested.

Unknown Signal #	Model's Prediction	Similarity to Circle (%)	Similarity to Square (%)
1	squared	5	95
2	squared	13	87
3	squared	4	96
4	squared	8	92
5	squared	4	96
6	squared	21	79
7	squared	5	95
8	squared	4	96
9	squared	7	93
10	squared	5	95

Table 1. Some of the predictions of the model

Ongoing Research

The remarkable achievement with the 2D fluid flow cases led us to move forward and use the CNN model for a utility-scale wind farm. Two different cases were selected to train the model. In the first wind farm, the turbine is aligned with the wind's direction (e.g., not yawed). In the second case, the turbine has a 10-degree deviation facing the westerly wind when seen from the top. The wind farm is a domain with length, width, and height of 2232 m, 1116 m, and 1000 m, respectively.

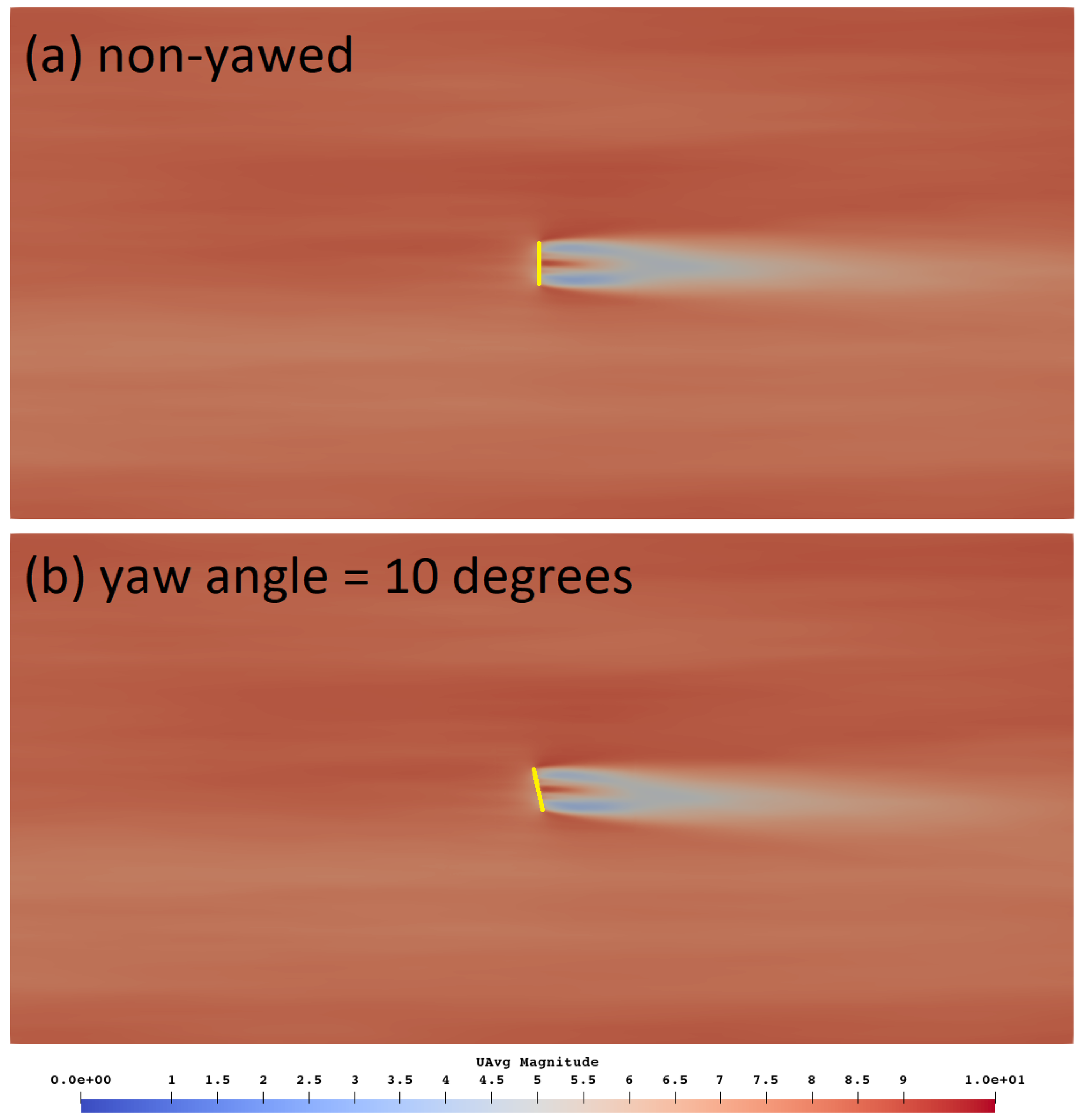

Figure 4. The utility scale wind farm seen from top at hub height. Case (a) is non-yawed and in (b), the turbine is rotated by 10 degrees .

Ongoing Challenges

Unlike the 2D case, the three-dimensional fluid flow in a wind farm is more turbulent. The impact of wind turbines could disappear quickly in the wake region. The areas in which the velocity signals are recorded must be selected wisely as the 10-degree yaw angle might not change the frequency of the flow noticeably, which makes it more challenging to train the model in turn. Another challenge is the expensive CFD simulation needed for each case. For instance, the mesh for the wind farm in Fig. 4 had about 6,000,000 cells in total. The current accuracy of the model in the wind farm application is 67%.

Conclusion

- GoogLeNet is a promising tool to predict the upstream event in the field of fluid mechanics. For a 2D flow, the model was applied successfully and predicted the unknown geometry.
- The application of the model on a utility-scale wind farm was considered. As the flow is entirely turbulent and the domain is 3D, more research is needed to achieve an acceptable accuracy and correct prediction.

Acknowledgments: The Center for Energy Systems Research at Tennessee Tech; Tennessee Tech's Information Technology Services for providing them with the high-performance computer cluster; The U.S. National Renewable Energy Laboratory (NREL) for providing and supporting the SOWFA package