29th International Conference
ENGINEERING MECHANICS 2023
Milovy, Czech Republic, May 9 – 11, 2023

INVESTIGATION OF THE VORTEX-INDUCED VIBRATION BY CONVOLUTION NEURAL NETWORK

Bublík O.[*], Heidler V.[**], Pecka A.[***] Vimmr J.[†]

Abstract: *The focus of this paper is to use a convolutional neural network to predict the unsteady laminar flow field in 2D domains, specifically as it relates to vortex induced vibration. In this scenario, an incompressible fluid is flowing around a cylinder, creating a vortex street that leads to oscillation of the cylinder. The movement of the cylinder, which has only one degree of freedom, is modeled using a linear spring-mass-damper equation. The research examines the cylinder's amplitude response for different spring stiffnesses and damping coefficients, and compares the results to those obtained from FSI computational fluid dynamics simulations.*

Keywords: Convolution neural network, Unsteady fluid flow, Vortex-induced vibration, Fluid-structure interaction.

1. Introduction

In this paper, we aim to use Convolutional Neural Networks (CNNs) to solve fluid-structure interaction (FSI) problems, which can cause structural failures due to phenomena such as flutter, vortex-induced vibration, or buffeting. These FSI problems are of great interest in fields like nuclear industry, aeronautics, and turbomachinery. However, simulating FSI problems is computationally expensive, especially when it comes to modeling fluid flow. To tackle this challenge, we propose to use a CNN to predict fluid flow instead of a traditional CFD solver.

CNNs were initially developed for image recognition (Fukushima, 1980), but they have since been used to predict fluid flow (Guo, 2016; Hennig, 2017). In these early studies, the authors showed that CNNs can predict flow fields significantly faster than CPU-based or GPU-accelerated CFD solvers, while maintaining an error below 3%. More recent studies (Bhatnagar, 2019; Thuerey, 2020) have further developed CNN models capable of predicting velocity and pressure fields around aerofoils of different shapes under varying flow parameters, such as Reynolds number and angle of attack.

While previous studies have focused on predicting fluid flow with stationary boundaries, our goal is to apply CNNs to FSI problems with moving boundaries.

2. Neural network architecture and training dataset

To predict the flow of unsteady incompressible fluid around a moving object, we used a U-Net (Ronneberger, 2015), a type of convolutional encoder-decoder neural network, as shown in Figure 1. The input to the network is a 3D array of size $128 \times 32 \times 8$ which includes information about the grid points at two different times (t_n and t_{n+1}), their x and y coordinates, boundary information, fluid velocity components (u and v), and pressure p at time t_n. The grid point coordinates are included because the mesh is non-Cartesian, and their positions change as the mesh deforms. The boundary information is a binary value that indicates whether the grid point is on the blade profile boundary or within the fluid domain. The output of the network is the pressure and velocity fields evaluated at the grid points at time t_{n+1}.

[*] Ing. Ondřej Bublík, Ph.D.: Faculty of Applied Sciences, University of West Bohemia, obublik@kme.zcu.cz
[**] Ing Václav Heidler, PhD.: Faculty of Applied Sciences, University of West Bohemia, vheidler@kme.zcu.cz
[***] Mgr. Aleš Pecka: Faculty of Applied Sciences, University of West Bohemia, pecka@kme.zcu.cz
[†] Assoc. Prof. Jan Vimmr, PhD.: Faculty of Applied Sciences, University of West Bohemia, jvimmr@kme.zcu.cz

ISSN: 1805-8256, doi: 10.21495/em2023-47

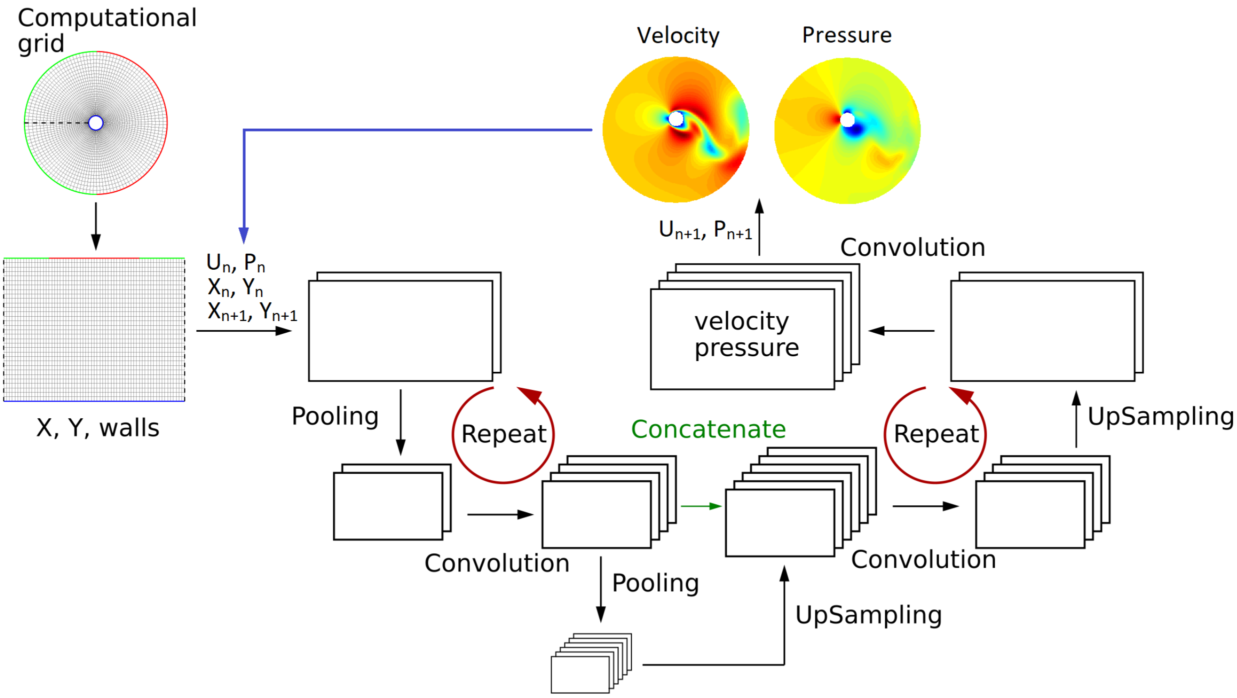

Fig. 1: Ilustration of the network architecture.

To accurately depict the flow near the object, we selected a circular domain with a tightly meshed area near the wall, as shown in the left part of Figure 2. However, Convolutional Neural Networks (CNNs) are designed for grids in a rectangular format, not for O-shaped grids. To resolve this, we transformed the mesh into a rectangular format by cutting it in front of the object and laying it out flat, as shown in Figure 2. The newly created boundary was assigned a periodic boundary condition, while the rest of the boundary was designated as open and served either as input or output.

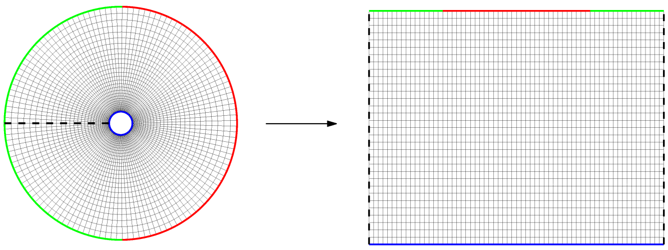

Fig. 2: Mapping of O-shaped mesh onto a Cartesian grid suitable to be processed by a CNN. The boundaries, namely inlet (green), outlet (red), wall (blue) and boundary with periodic boundary condition (dashed black), are highlighted.

3. Structure model and fluid-structure interaction

We study the interaction between a laminar fluid flow and a rigid body that is mounted elastically and restricted to move horizontally in a 2D cross-section. The dynamics of the elastically-mounted body is described by a 1-DOF linear mass-spring-damper model

$$\ddot{y} + 2\,\zeta\,\omega_n\,\dot{y} + \omega_n^2\,y = \frac{L}{m}. \tag{1}$$

Here y is displacement in the horizontal direction, $\zeta = \frac{c}{2m\omega_n}$ is the damping ratio, $\omega_n = 2\pi f_n = \sqrt{k/m}$ is the undamped angular natural frequency, m is mass, c and k are the damping and stiffness coefficients and L is the lift force. The damped natural frequency is related to the undamped natural frequency as follows

$$f_d = f_n\sqrt{1-\zeta^2}. \tag{2}$$

We discretize this equation using the BDF2 method, which is a 3-level implicit method.

To ensure accurate fluid-structure interaction, two requirements must be met at the fluid-structure boundary: balance of forces and geometric continuity. Under the assumption that only aerodynamic forces are acting on the body, the balance of forces condition is established

$$L = \oint_{\Gamma} \left(\sigma_{xx}\, n_x + \sigma_{yx}\, n_y\right) \mathrm{d}S. \tag{3}$$

Here Γ is the boundary of the body, n is the unit outer normal to the boundary and σ is the aerodynamic stress tensor.

To summarize, the FSI solver integrates a CNN that predicts fluid flow with a differential scheme that handles the structure's dynamics. The fluid flow solver based on CNN and the structure solver are weakly connected. The continuity of the fluid-solid boundary is maintained by adjusting the fluid domain in response to the movement of the rigid body.

4. Numerical results

After training a CNN on the unsteady flow field around a cylinder undergoing prescribed harmonic motion with varying amplitudes and frequencies, we integrated the CNN with a structure solver to conduct FSI simulations. To validate the accuracy of the CNN-based FSI solver, we compared its results to those of a CFD-based FSI solver. Both solvers utilized the same structure solver and coupling algorithm, the only difference being the fluid solver. We utilized FlowPro (Bublik, 2000) as the fluid solver, which was also used to generate the training dataset.

To assess the accuracy of the CNN-based FSI solver, we conducted simulations with varying natural frequencies and damping ratios of the structure. The flow conditions remained the same in all simulations, including a Reynolds number of 100. The damping ratios used were 0.375 and 0.45. For each damping ratio, simulations were run with different natural frequencies of the structure and the resulting amplitudes and frequencies were plotted. Fig. 3 displays a comparison between the amplitude characteristics obtained from the CNN-based and CFD-based FSI solver for specific damping ratios, indicating the amplitude response of the cylinder near the resonant natural frequency. The damped natural frequency f_d is normalized by the Strouhal frequency f_{St} (the vortex-shedding frequency for a fixed cylinder), and the amplitude A is normalized by the cylinder's diameter D. Both f_{St} and D are constant across all simulations.

Fig. 4 presents the relationship between the steady-state oscillation frequency f of the cylinder and the damped natural frequency f_d of the structure, with both values normalized by f_{St}. The horizontal black line represents the Strouhal frequency f_{St}, which has a value of 1 due to normalization. The inclined black line represents the natural frequency of the cylinder f_d, and has a slope of 1, since $y = x$ after normalization. The blue circles depict the frequencies obtained from the CFD-based FSI solver, while the red squares represent the frequencies obtained from the CNN-based FSI solver. When the natural frequency is close to the Strouhal frequency $f_d/f_{\mathrm{St}} \approx 1$, the vortex-shedding frequency adjusts and begins to follow the natural frequency of the structure. This results in a deviation from the horizontal line and towards the inclined line in the vicinity of resonance, and is referred to as the lock-in phenomenon.

5. Conclusions

In this study, a convolutional neural network (CNN) is employed to predict unsteady laminar flow with a moving boundary. The boundary of the fluid domain is traced using a deformable non-Cartesian grid. The CNN was trained on simulations of an oscillating cylinder at different frequencies and amplitudes. The motion of the elastically-mounted cylinder was modeled using a linear spring-mass-damper system, which was solved using an implicit differential method. The results showed that the CNN-based FSI solver accurately captured the "lock-in phenomenon" in the vortex-induced vibration of a cylinder, producing results similar to those from a CFD-based FSI solver. Additionally, the CNN solver was found to be two orders of magnitude faster than the CFD-based solver, and this speed advantage is expected to increase for larger problems.

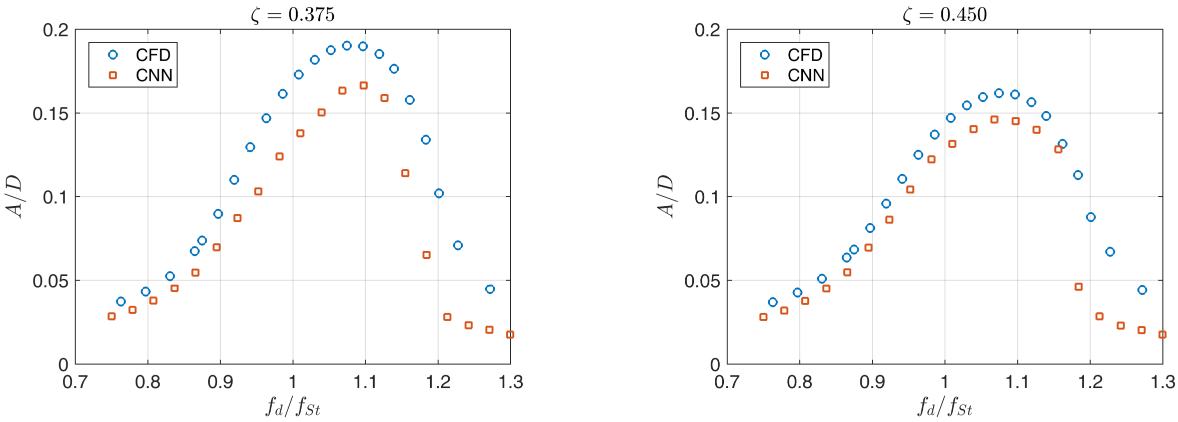

Fig. 3: Cylinder amplitudes A depending on the damped natural frequency f_d for various damping ratios ζ. The blue circles are the CFD-based results while the red squares are the CNN-based results.

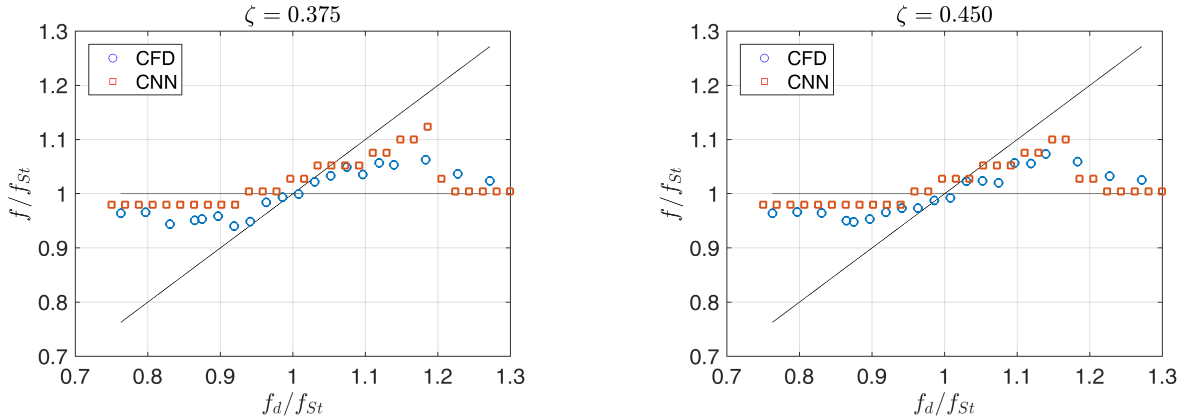

Fig. 4: Steady-state oscillation frequency f depending on the damped natural frequency f_d for various damping ratios ζ. The blue circles are the CFD-based results while the red square are the CNN-based results.

Acknowledgments

This research is supported by the projects GA21-31457S ”Fast flow-field prediction using deep neural networks for solving fluid-structure interaction problems”.

References

Bhatnagar, S. and Afshar, Y. and Pan, S. and Duraisamy, K. and Kaushik, S.(2019), Prediction of aerodynamic flow fields using convolutional neural networks, *Computational Mechanics*, Vol 64, No.2, pp 525-545.

Bublík, O. and Pecka, A. and Vimmr, J. (2000), FlowPro - multipurpose CFD software written in Java, *Proc. of Computational Mechanics*, University of West Bohemia.

Fukushima, K. (1980), Neocognitron: A self-organizing neural network model for a mechanism of pattern recognition unaffected by shift in position, *Biological Cybernetics*, Vol 36, No.4, pp 193-202.

Guo, X. and Li, W. and Iorio, F. (2016), Convolutional neural networks for steady flow approximation, *Proc. the ACM SIGKDD International Conference on Knowledge Discovery and Data Mining*, pp. 481.

Hennigh, O., (2017), Lat-Net: Compressing Lattice Boltzmann Flow Simulations using Deep Neural Networks.

Ronneberger, O. and Fischer, P. and Brox, T. (2015), U-net: Convolutional networks for biomedical image segmentation, http://arxiv.org/abs/1505.04597.

Thuerey, N. and Weißenow, K. and Prantl, L. and Hu, X. (2020), Deep learning methods for reynolds-averaged navier–stokes simulations of airfoil flows, *AIAA Journal* Vol 58, No.1, pp 25–36.