Journal of Physics: Conference Series

PAPER • OPEN ACCESS

CFD analysis of the complex japanese pufferfish nest

To cite this article: Jialan Dai *et al* 2024 *J. Phys.: Conf. Ser.* **2741** 012010

CFD analysis of the complex japanese pufferfish nest

Jialan Dai[1], Hossein Zare-Behtash[1, 3] and Chuanbo Dai[2]

[1] James Watt School of Engineering, University of Glasgow, Glasgow, G12 8QQ, United Kingdom
[2] Jiaying University, Meizhou, 514000, China

[3] Corresponding author's e-mail: Hossein.Zare-Behtash@glasgow.ac.uk

Abstract. Existing research on the pufferfish species (Torquigener Albomaculosus) has shown that its nest structure has a certain impact on flow velocity. This study aims to investigate the nest-building behavior by using different materials. A 3D model will be created based on nest geometry, and 2D simulations will be conducted by using computational fluid dynamics (CFD). Furthermore, the study examines the nest's fluid dynamics characteristics in its original state, with varied materials and heights. Results demonstrate that the nest reduces flow velocity, adapts to external flow conditions, and enhances shear strength with rough materials. Additionally, nest height affects internal circulation area and pressure. This unique structure protects the spawning area, improves oxygen transport, and strengthens the nest. The study further expands biomimetic engineering applications and suggests future interdisciplinary research directions.

1. Introduction

Biomimetics has played a significant role in engineering. Examples include collision avoidance systems inspired by bamboo structures, reduction of turbulent drag using shark skin structures, and satellite folding solar wings inspired by insect wing designs [1-3]. Moreover, animal nest-building behavior serves for communication and offspring protection, with potential engineering applications [4]. In this study, CFD technology will be used to investigate the flow field characteristics of a pufferfish nest, a new species discovered in 2013. Compared to laboratory simulations, CFD has advantages in terms of convenience, economy, and visual representation [5].

The "Mystery Circles" phenomenon, discovered in 1995 near subtropical islands in Japan, features circular structures with wave-like protrusions and irregular patterns in the center. The circular shape guides water flow and accumulates fine sand for egg deposition [6]. Previous research has shown that pufferfish nests reduce flow velocity and enhance shear strength [7]. However, most studies have used smooth and streamlined nest models, leaving the investigation of nest-building behavior with different materials open. This study explores the impact of rough materials, simulating fluid dynamics with shell-shaped protrusions by using StarCCM+. It analyzes nest geometry's influence on the internal flow field and proposes potential applications in functional engineering.

The nest and pufferfish's unique building behavior are shown in Figure 1.

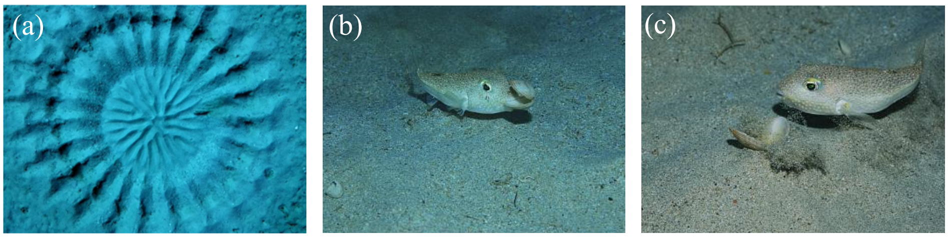

Figure 1. (a).The completed nest; (b),(c).The behavior of pufferfish constructing nests by using shells. (Adapted from BBC One, 2015; Nature Picture Library, 2017)

2. Methods

The main workflow of this study is illustrated in Figure 2.

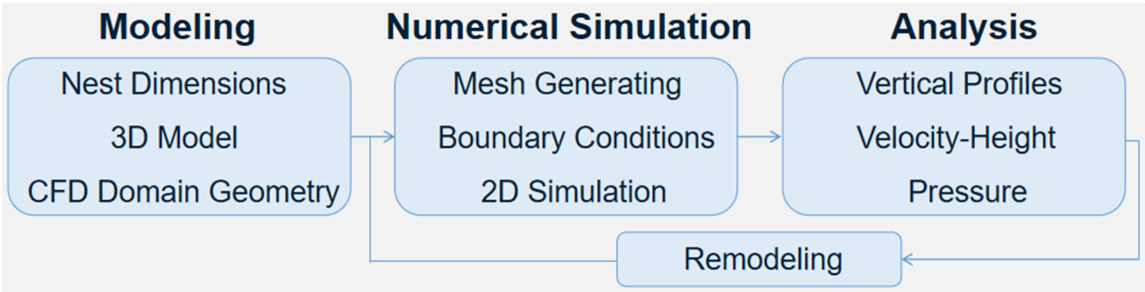

Figure 2. The workflow of this project for modeling and CFD simulation.

2.1. Construction of nest geometry

Firstly, the pufferfish nest's geometry is highly complex, consisting of a circular outer ring and irregular patterns within it. The 3D modeling process of the pufferfish nest is a critical step in this study.

Referring to the nest's geometry in Figure 1, the modeling ensures the maximum resemblance to the nests constructed by the pufferfish. Firstly, the pufferfish nest's geometry is highly complex, consisting of a circular outer ring and irregular patterns within it. Therefore, obtaining accurate data is crucial for the modeling process. Pufferfish dedicate nine days to construct their nests, gradually expanding the structure from around 600 mm in radius to 918 mm [8]. The number of valleys increases while the center area becomes flatter. Additionally, the height variation between radii and peaks also increases. On the last day, the center of the "Mystery Circles" presents irregular patterns, and the radial valleys split into two peaks. The outer surface of the outer ring is adorned with fragments of shells and corals. In addition, digital image measurement techniques can also be used to model the nest. Digital photogrammetry techniques allow accurate modeling of 3D space from 2D images [9]. Based on existing images, digital photogrammetry techniques are used for the 3D reconstruction of nest images [7]. After comparing the data and information, it is found that the results of this modeling technique are satisfactory.

To achieve precise 3D modeling, the Computer-Aided Design (CAD) software, Autodesk Inventor, will be used in this study to create the geometric model of the pufferfish nest based on collected data. The software will depict the circular structure of the outer ring and intricately represent the internal irregular patterns, recreating the nest's complex features.

Based on the exciting research [7, 9], the entire outer circular radius of the nest is set at 918 mm, and the radius of the center region (living area) is set at 306 mm (one-third of the outer circular radius). As the center region is lower than the seabed, a reference plane is set below the seabed by 40 mm. The height of the first peak is set at 140 mm, and the second peak at 89 mm. The specific nesting dimensions are shown in Table 1.

Table 1. Nest Dimensions.

	Outer Ring	Central Area	First Peak	Second Peak	First Valley	Second Valley
Radius (mm)	918	306	800	490	573	306
Height (mm)	Seabed 40	20	140	89	65	10

The number of ridges is set to 30, resulting in the complete pufferfish nest model (Figure 3).

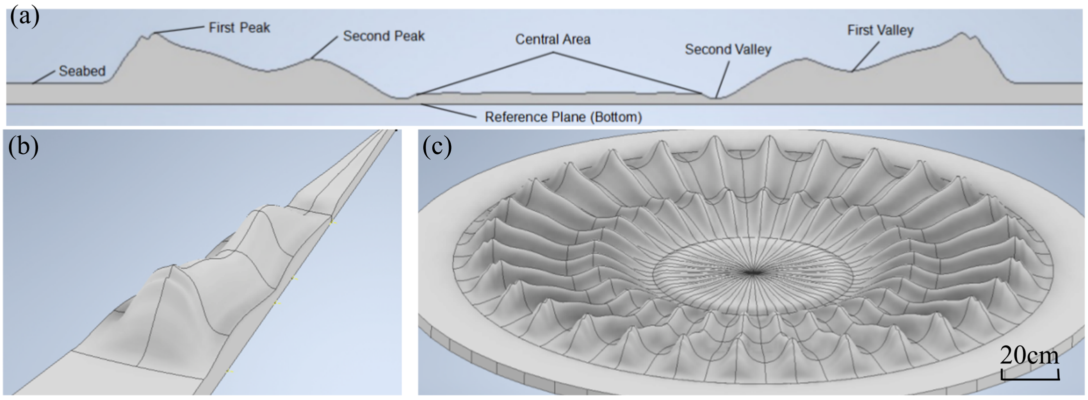

Figure 3. The 2D cross-section and 3D model of the nest; (a).Cross-section; (b),(c).3D geometry.

This study will investigate the purpose of pufferfish placing shells outside the nest's outer ring and the impact of different materials on the overall structure and flow field. The original model will be modified by adding two different sizes of shell-shaped protrusions to the surface (Figure 4). The outer ring of the nest will be divided into three parts: Front End (FE), Anterior Peak (AP), and Posterior Peak (PP) surfaces. Separate modeling will be conducted for the shells at these locations, resulting in four sets of models. Each position will be studied individually, and then the flow field of the complete outer ring with all the added elements will be investigated.

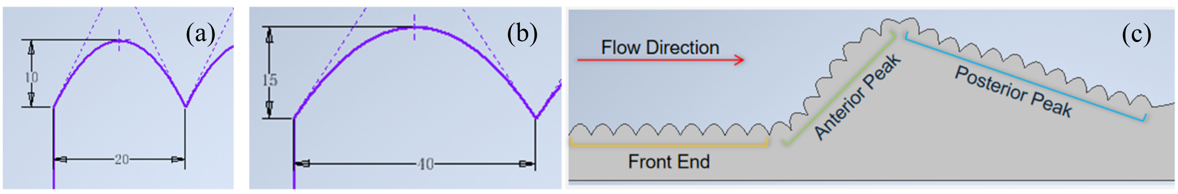

Figure 4. (a),(b). Add-on shell dimensions; (c).Shells (large) on three surfaces.

In addition, StarCCM+ can only simulate 3D models. To perform 2D simulations on the model, it is necessary to radially slice and stretch the 3D model, and then import the custom 2D computational domain into StarCCM+.

2.2. Numerical simulation

First of all, it is worth noting that this study focuses primarily on the influence of the nest's geometric shape on the flow field. Therefore, the materials of the nest and the distribution of internal stress are not within the scope of the investigation.

When conducting numerical simulations, certain characteristics of water are often simplified to reduce computational complexity. This study considers the following factors: treating seawater as a single continuous medium, neglecting molecular movements; neglecting viscosity due to low flow velocities [10]; neglecting temperature and salinity gradients in typical marine environments [11]; setting a simplified initial flow velocity of 0.8 m/s; assuming nearly constant density and employing incompressible Navier-Stokes equations [12]:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

where u_i is the corresponding velocity components, x_i is Cartesian coordinates, p is pressure, t is time, and τ_{ij} is the turbulent stress. Simultaneously, this study utilizes the finite volume method and employs 2D simulation to model the pufferfish nest's flow field, which provides computational simplicity, reduces resource requirements, and offers clearer visualization. The Reynolds-Averaged Navier-Stokes (RANS) model with the SST (Menter) K-Omega model is employed for accurate near-wall and far-field flow resolution [13]:

$$T_t \equiv -\rho\overline{v'v'} = -\rho\begin{bmatrix} \overline{u'u'} & \overline{u'v'} & \overline{u'w'} \\ \overline{u'v'} & \overline{v'v'} & \overline{v'w'} \\ \overline{u'w'} & \overline{v'w'} & \overline{w'w'} \end{bmatrix} \tag{3}$$

To establish the computational domain, dimensions, and boundary conditions are determined. The basic dimensions of the nest are approximately 1.8 m*0.15 m. Regarding this, the domain size is set as Table 2.

Table 2. Domain Dimensions.

	Height	Inlet from Front	Outlet from Rear
Dimensions (m)	7.5	15	30

Also, the overall configuration of the computational domain is illustrated in Figure 5.

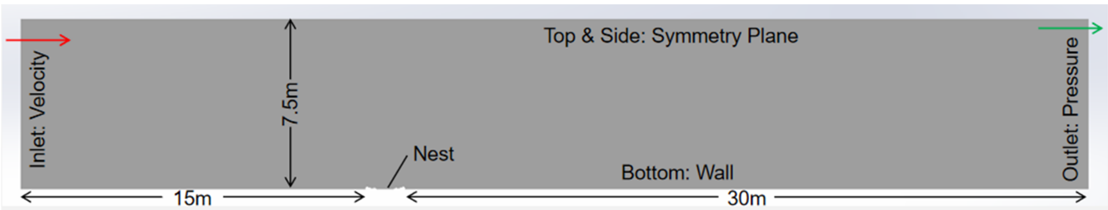

Figure 5. Computational domain with boundary conditions.

Grid generation employs finer mesh near the nest and coarser mesh in the far field as shown in Figure 6.

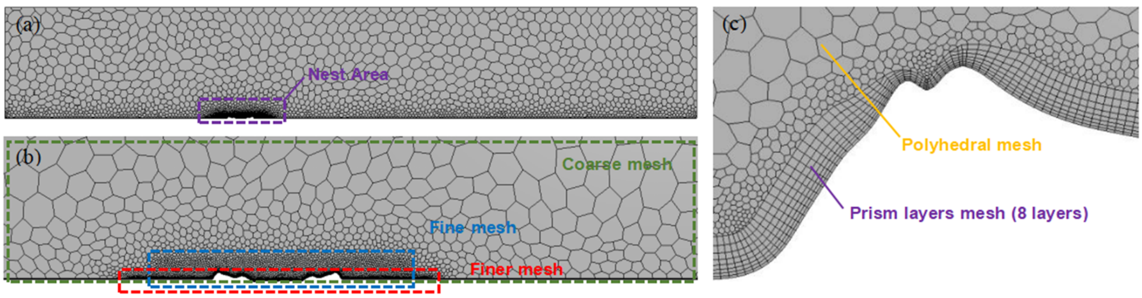

Figure 6. Meshing strategy - finer mesh near the nest. (a). Complete computational domain; (b),(c). Meshing strategy near the nest area and the surface.

Convergence is determined by examining the steady state density ratio, turbulent kinetic energy, and other curves in the residual chart. In this study, with 5000 iteration steps and residuals below 10^{-5}, good convergence is achieved, confirming mesh quality and the simulation accuracy.

2.3. Flow characteristic evaluation

The evaluation of the flow field within the nest is conducted based on the following aspects:

- Examining the flow characteristics by analyzing the velocity and pressure distributions.
- Investigating the influence of nest height on the flow field by observing changes in the velocity field when altering the overall nest height.
- Assessing the deceleration effect by generating height-velocity relationship curves using vertical probes at different positions within the nest.
- Reflecting the strengthening effect of the structure and the presence of additional shells by analyzing the distribution of shear stresses on the nest's surface and the seabed surface.

3. Results

3.1. Initial circulation zone

The original simulation is conducted with a velocity probe at the nest's central region. It is observed that the bottom exhibits reverse flow (min. velocity: 0.186 m/s), increasing gradually. Velocity drops to 0 at 0.112 m above the bottom and peaks at 0.879 m/s at 0.380 m. The velocity scalar field is generated with an inlet velocity of 0.8 m/s (V_0) for internal flow observation.

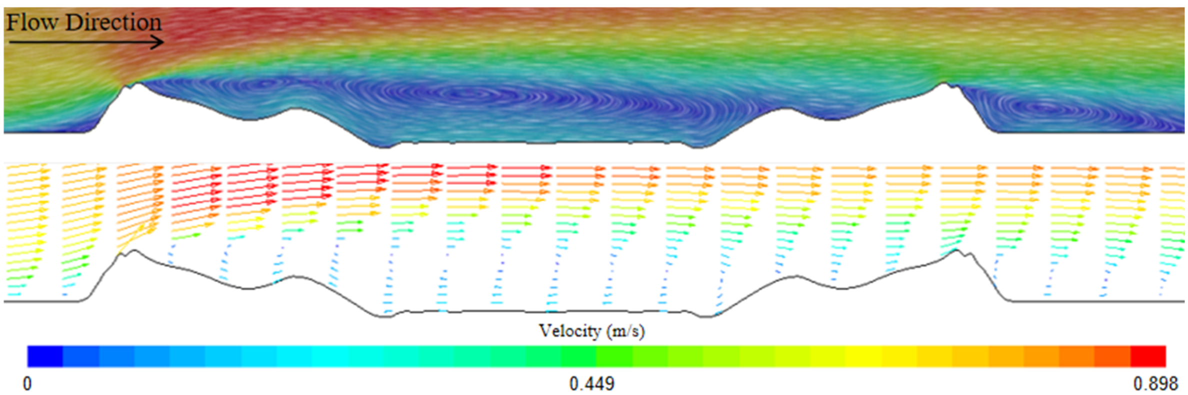

Figure 7. Velocity contour plot of the 2D cross-section and velocity vectors.

From Figure 7, the water flow accelerates as it passes through the first peak, forming a dome-shaped structure above the nest with a high velocity, and the edge of the dome has a rapid velocity, which decreases to below 0.5 m/s. Inside the nest, a low-speed and clockwise-rotating recirculation area forms. It is noteworthy that the velocity near the bottom is around 0.1-0.2 m/s. The final section junction has a turbulent region, gradually smoothing to initial flow velocity.

Furthermore, when the initial flow velocity changes to 0.4 m/s, 1.2 m/s, and 1.4 m/s, the nest displays consistent deceleration and merging of small and central turbulence. Overall, flow position and velocity magnitude remain relatively unchanged across varying velocities.

Figure 8 shows the pressure distribution, with the lowest pressure occurring at the two peaks. These results are consistent with the analysis using Bernoulli's principle in the velocity field.

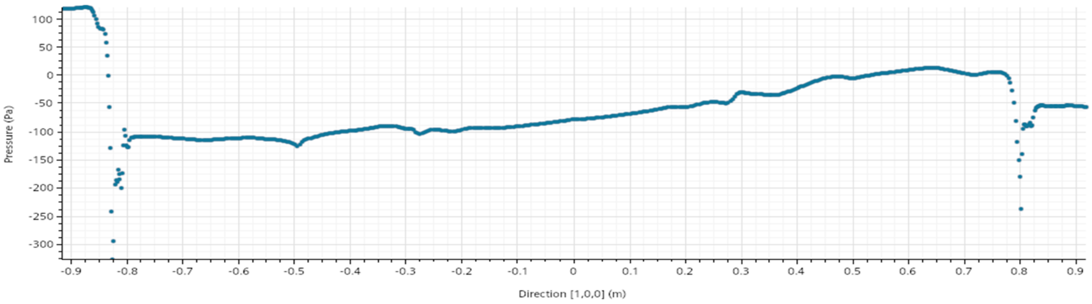

Figure 8. Pressure distribution along the inner wall of the nest (dashed line).

Additionally, varying the height of the first peak alters the internal flow (Figure 9). A higher peak leads to the merging of turbulence and backward shift. When it is beyond the height threshold, counter-flowing turbulence appears. Meanwhile, the dome still covers the entire nest, and the flow velocity inside the living area remains below half of the initial flow velocity.

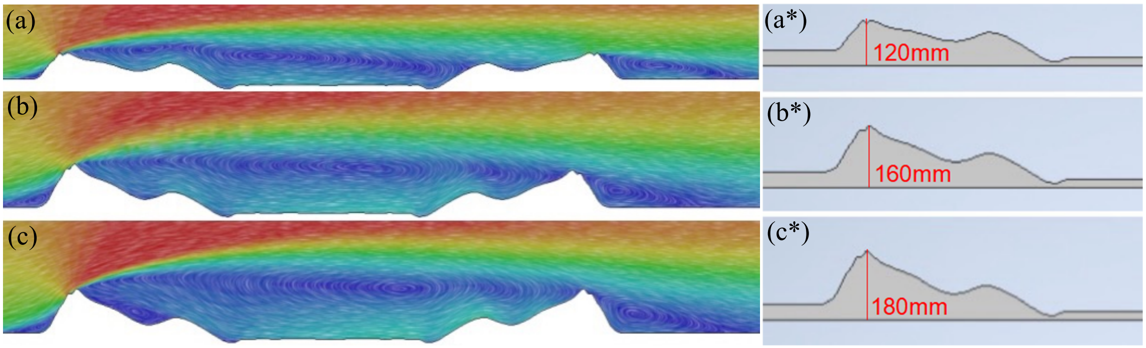

Figure 9. The velocity field under different heights. (a).120 mm; (b).160 mm; (c).180 mm.

3.2. Adding shell-shaped protrusions

After remodeling and simulation, the changes generated by adding shells in a single region are limited. By adding vertical profiles and shells in all locations (FE, AP, PP), the curves obtained are shown in Figure 10.

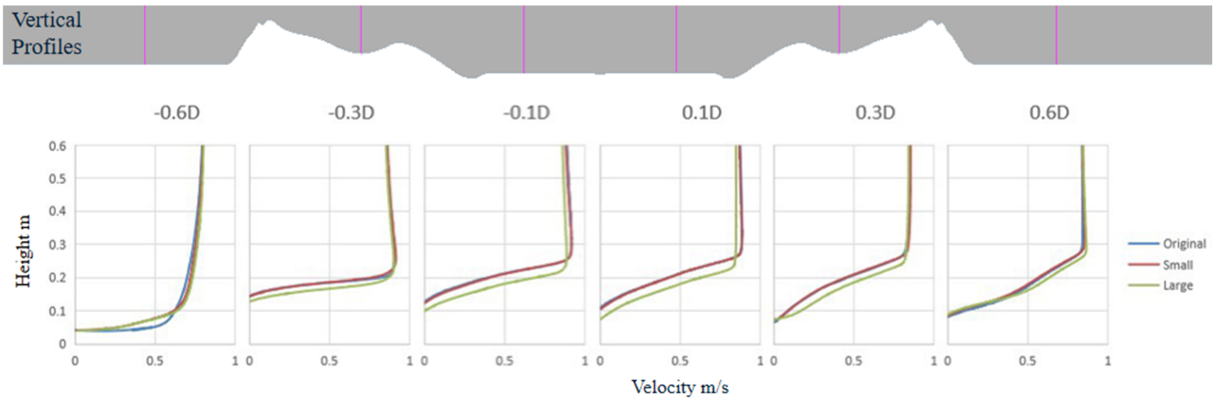

Figure 10. Vertical profiles and velocity-height curve with all protrusions.

It can be observed that the large shells (green curves) result in a maximum velocity that is consistently lower than the average values of the other two groups. The deceleration effect inside the nest is more pronounced. At the same time, the height corresponding to the same velocity is also reduced. The deceleration effect is more significant near the bottom of the nest. In contrast, the deceleration effect of the small shells on the water flow is relatively limited. After calculating the specific data, it is found that the average velocity in the central area of the nest, with the addition of large shells (Figure 11), decreases by approximately 4.42%.

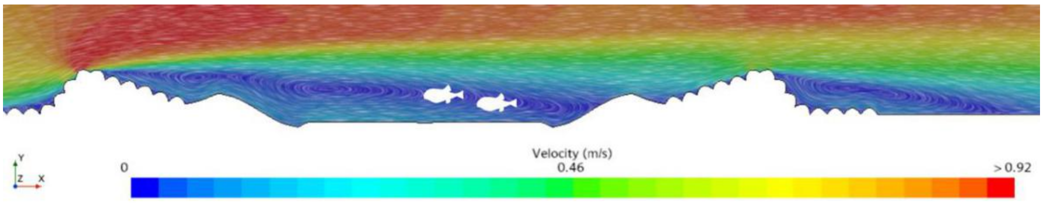

Figure 11. Pufferfish inside the nest

From Figure 12, the maximum shear stresses at peak regions reach the maximum, and in the central area with fine sand, those remain at a lower level. Respectively, it is confirmed that the rough material enhances the shear strength of the nest and provides a comfortable living environment.

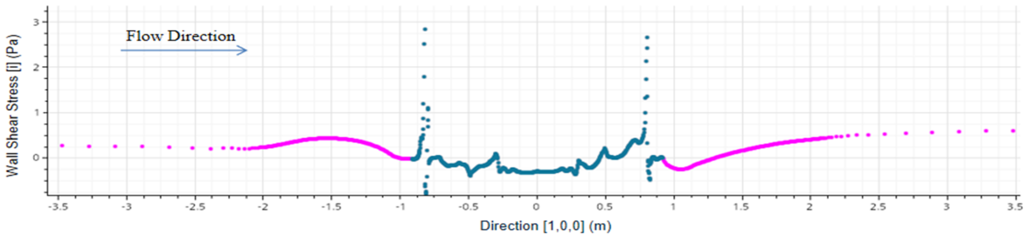

Figure 12. Wall shear stress distribution along the bottom (pink-seabed area; blue-nest area).

4. Discussion

As mentioned in the previous chapter, the nest structure effectively reduces the flow velocity and enhances shear strength, providing low-energy consumption, and a secure and stable hatching environment for their living and reproductive behaviors.

Based on its flow control mechanisms and material selection strategies, the following possible engineering approaches are proposed:

- Construction of fish shelters and aquaculture facilities: reducing the survival pressure and abnormal behavior frequency of fish. Also, it is necessary to provide sufficient structural strength and rigidity while minimizing manufacturing costs.
- Pressure regulation in underwater applications: low-pressure regions for sensitive devices.
- Coastal protection and erosion control: artificial structures with wave-like configurations to reduce the impact of strong currents and storms, protecting coastlines from erosion and maintaining coastal stability (Figure 13).

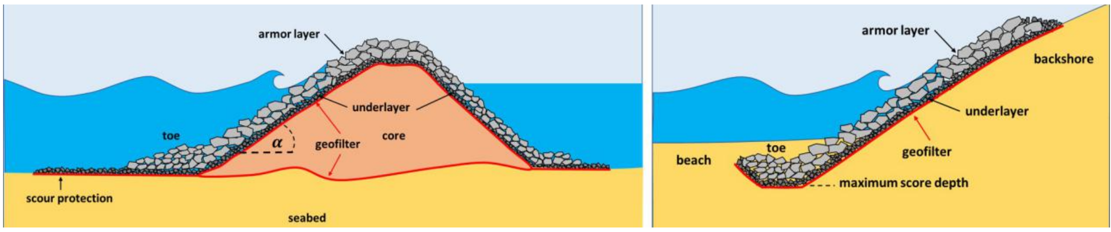

Figure 13. Offshore breakwater.
(Adapted from Dronker, Coastal Wiki, 2020)

- Innovative building materials: improving the stability of infrastructure exposed to fluid forces.
- Terrestrial biomimetic architecture: further interdisciplinary collaboration is needed.

5. Conclusions

This study conducts CFD analysis on the complex geometric structure of the pufferfish nest and its unique nest-building methods. By varying the geometric features, initial flow velocities, and height of the nest, the variations in velocity profiles, pressure fields, and shear stresses are further analyzed. This study reveals that:

- The nest has a noticeable decelerating effect on the flow velocity in the central region.
- The rough outer ring area can further enhance the deceleration effect and improve the shear strength of the structure.
- The height of the first peak significantly affects the internal flow field. The best deceleration effect is achieved when the height ratio is approximately 0.08.
- Different flow velocities generally do not alter the internal flow field, and the deceleration effect of the structure is similar.
- The nest structure can be used for engineering problems involving deceleration at the center of a circular region.

Limitations:

- Due to computational constraints, 2D simulation was conducted instead of 3D.
- Only the fluid characteristics of the structure in water were studied, and no other media were investigated.
- The nest shape was only modeled based on reference data and video materials. By using image terrain reconstruction techniques, a more accurate model can be obtained.

References

[1] Zou, M., Song, J., Xu, S., Liu, S., & Chang, Z. (2018). The bionic design of the bumper beam inspired by the bending and energy absorption characteristics of bamboo. *Applied Bionics and Biomechanics, 2018.*

[2] Dean, B., & Bhushan, B. (2010). Shark-skin surfaces for fluid-drag reduction in turbulent flow: a review. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 368*(1929), 4775-4806.

[3] He, J., Zhang, Y., Shangguan, Z., & Yang, L. (2020, June). A Review of Bionic Design in Satellite Solar Wing Structures. In *Journal of Physics: Conference Series* (Vol. 1549, No. 4, p. 042099). IOP Publishing.

[4] Hansell, M. H. (2005). *Animal architecture*. Oxford University Press, USA.

[5] Anderson, J. D., & Wendt, J. (1995). *Computational fluid dynamics* (Vol. 206, p. 332). New York: McGraw-Hill.

[6] Kawase, H., Okata, Y., & Ito, K. (2013). Role of huge geometric circular structures in the reproduction of a marine pufferfish. *Scientific reports*, 3(1), 1-5.

[7] Shameem, A. J., Valyrakis, M., & Zare-Behtash, H. (2021). A Fluid Dynamics Approach for Assessing the Intelligent Geomorphic Design of the Japanese Pufferfish Nest. *Geosciences*, 11(1), 22.

[8] Kawase, H., Mizuuchi, R., Shin, H., Kitajima, Y., Hosoda, K., Shimizu, M., ... & Kondo, S. (2017). Discovery of an earliest-stage "mystery circle" and development of the structure constructed by pufferfish, Torquigener albomaculosus (Pisces: Tetraodontidae). *Fishes*, 2(3), 14.

[9] Egels, Y., & Kasser, M. (2001). Digital photogrammetry. CRC Press.

[10] Turkel, E. (1996, June). Preconditioning methods for low-speed flows. In *14th Applied Aerodynamics Conference* (p. 2460).

[11] Mamayev, O. I. (2010). *Temperature-salinity analysis of world ocean waters.* Elsevier.

[12] Kwak, D., Kiris, C., & Kim, C. S. (2005). Computational challenges of viscous incompressible flows. Computers & fluids, 34(3), 283-299.

[13] Dewan, A. (2010). Tackling turbulent flows in engineering. Springer Science & Business Media.