Submarine channel initiation using realistic computational fluid dynamic simulation

A A Mujiyanto, O A Prasojo and T L Indra
Geoscience Study Program, Faculty of Mathematics and Natural Sciences (FMIPA), Universitas Indonesia, Depok 16424, Indonesia
Corresponding author’s email: tito.latif@ui.ac.id

Abstract. Submarine channel initiation is the process that involved turbulence fluid flow and sculpting submarine channels. The fundamental factor that affects this process is fluid flow velocity and grain-size deposit that the fluid itself carry. This study presents a realistic computational fluid dynamic simulation to simulate the process of submarine channel initiation in a time-dependent equation. This study uses a standard slope model but with the novel approach of adjusted parameters of the actual condition of the submarine channel initiation such as depth hydrostatic pressure, fluid density, and fluid velocity. The result of the simulation showed the correlation between fluid flow velocity and deposited grain-size distribution, which resulting bigger grain size are deposited along with higher fluid flow velocity at the highest activity of 14 m/s and smaller grain size are deposited at lower fluid flow velocity at the lowest activity of 0 m/s among the geometry of the submarine fan.

Keywords: Submarine, channel, turbulence, simulation

1. Introduction
Submarine channels are flume for sediment-gravity flows that carve continental margins as it brings sediment into the deep water [1]. Sediment-gravity flows are mixtures of water and sediment where the sediment part pulls the occupying water downslope along with the force of gravity [2]. Submarine channels are essential components in the deep water fans which consist of canyons, channels, levees, and depositional lobes [3]. Submarine channels can extend across the seafloor for several hundred to thousands of kilometers long [4]. In these submarine channels, the deposits can host significant hydrocarbon resources [5].

Submarine channel initiation is part of the submarine channel evolution [6]. The evolution is the result of dynamic interaction between turbidity currents and the seafloor [7]. However, the morphodynamic properties are rarely directly observed in the seafloor [8]. Hence many geoscientists used the experimental approach to simulate the submarine channel evolution, including the initiation process. Several numbers of physical experiments have been successfully produced subaqueous channels using a saline flow over a mobile substrate [9-11]. The result of these previous studies provided great insight into the submarine channel initiation. But the majority of the physical experiments mostly have been made in above sea level atmospheric pressure such as on ground laboratory which is neglecting the atmospheric pressure parameter.

In this paper, we presented the experiment through Computational Fluid Dynamic (CFD) simulation to simulate a nearly identical condition in the submarine channel initiation process.
The simulation was achieved by creating geometry and scaling down the parameters which abide two focuses, (1) Navier-Stokes fluid dynamic interactions and (2) Reynolds number parameter.

2. Data and method

2.1. Experimental setup

The experimental setup used a cross-platform finite analysis software called COMSOL Multiphysics® for the computational fluid dynamic simulation. The software provided ready-made physics interfaces that can receive model input and necessitate sophisticated fluid flow models regarding the submarine channel initiation which involves the single-phase turbulence flow under the influence of gravity and hydrostatic pressure.

The geometry constructed from two blocks assembled in such a manner to represent both the slope and the seafloor. Each of the geometry sides was configured to match the natural condition in the deep water zone. The size of the geometry conducted as the main box is a square of 20 m × 20 m with a total height of 5 m, and the inlet is a tilted box with the size of 5 m × 5 m. The inlet is assembled in tilted shape to provide better flow initiation as the objective is to simulate such flow driven by gravity force.

Figure 1 shows the side that configured as the inlet for the fluid flow (A). The sides that face upward (B) configured with hydrostatic pressure below of 3000 meters below the sea level. The geometry surrounded by the lateral side (C) that configured as continuity boundary. For each side at the protruding block (D) is the channel boundary side. At the bottom side of the geometry configured with the roughness of sand-quartz to mimic the deep water sediment surface.

2.2. Incompressible Navier-Stokes equation

This experiment used the single-phase turbulence flow in the process of simulation. The fluid dynamic interactions in the experiment must undergo through the identification of natural condition, an incompressible fluid field. The fluid is assumed to be isotropic and it does not depend directly on the flow velocity, but only on spatial derivatives of the flow velocity. This also applied in turbulence flow under the sea as the increase of the depth could make the fluid denser and pressurized.

Regarding the turbulence flow, it is the time-dependent that happen in chaotic behaviour seen in many fluid flows. However, the numerical solution of the equation is quite difficult due to significantly different mixing-length scales that are involved in turbulence flow. The experiment used Reynolds-averaged Navier Stokes Equations (RANS), supplemented with proper turbulence models which in this case used the Turbulence Kinetic Energy model (k-ε model). This approach resulted in less expensive in time and computer memory with a good result as it explicitly resolves a good enough number of iterations.

2.3. Scaling down approach

The scaling down calculation used the Froude Scale Model, given by

\[ Fr = \frac{U}{\sqrt{g'h}} \]  

Froude number \((Fr)\) is a nondimensional hydraulic variable that very important in describing flow mechanics as it dictates the propagation of waves within a fluid flow. Froude number is given by
Figure 1. The geometrical setup used in the CFD simulation. Here, A denotes the inlet side, B is the upward side, C is lateral side and D is channel boundary side. Figure scale is in the meter.

dividing the current velocity $U$ with a variable of experimental Boussinesq approximation $g'$ based on modified flow acceleration and current thickness $h$. In this case, should the inertial forces exceeded the gravitational force ($Fr > 1$), then the flow is tagged as supercritical. Vice versa, should the converse is true ($Fr < 1$), then the flow is tagged as subcritical where the waves are propagated upstream and downstream [11]. By using this approach, the hydraulic behaviour of the experiment will be maintained to remain similar to the natural setting by maintaining the same Froude number value both in the experiment and in the natural setting.

Besides that, the experiment also used another important nondimensional number Reynolds number, given by

$$Re = \frac{Uh}{\nu}$$

(3)

Reynolds number is the ratio of inertial forces to viscous forces and describes whether the flow is either laminar flow ($Re < 575$), transitional flow ($757 < Re < 2000$), or turbulent flow ($Re \geq 2000$) [12].

As the Reynolds number is a fixed number, the scaling down must follow the Reynold number parameter. Hence the most viable options are either to scale down the geometry of the submarine fan such as total area, total height, or scaling down the fluid velocity without changing the Reynold number to below 2000 and changing the Froude number to below 1.

3. Results and discussion

Here are the detailed results of the computational fluid dynamic simulation in submarine channel initiation by using the experimental setup and theoretical consideration as aforementioned. The turbulence flow set up by calculating from both equation 2 and equation 3 based on Hamilton et al. [13], it needed a minimum flow velocity of 7 m/s with Re above 3000 (indicating the flow is turbulent). The experiment used 10 m/s flow velocity as input flow, inlet side configured at side A of the geometry (figure 1). The fluid flow at inlet shall interact directly with the downslope surface to create a particle-laden mixture resulting in a gravity-driven turbulence flow. All the
downside surfaces are simulated with sand-quartz material from the COMSOL material library. After the turbulence flow passed through the slope, it triggered the submarine channel initiation. The color scale range from red to blue whereas indicate red as the field with higher velocity and blue as the field with lower velocity. From figure 2, we could see the range of color dispersion showing the velocity field that happened after the submarine channel initiation. The protruding block that represents inflow velocity would have higher velocity as it was the source of the flow. Then along the midway of the slope resulted in a teal-colored field which indicated similar velocity field as the inlet side. An important note should be taken that there is the reason why inlet side is not on red-colored, and that which is because the experiment considers that phenomena were assumed to be a simultaneous turbulence flow in an intermittent manner.

The result gave the pretty much-expected image as the highest velocity field is in the middle part of the geometry (figure 3). The pattern of the velocity field itself also showed a turbulence flow as the submarine channels initiated. The each of farthest side showed a relatively darker blue in color. The temporal teal-colored velocity field at the inlet was the result of the computational mechanism that assumed the fluid source were big enough to prompt the turbulence flow along the downslope. In contrary, the colour further up along the y-axis as it could be interpreted as either the turbulence flow is still interacting at the proximal side or the velocity field at that surrounding area is really low.

The present experiment confirms that there is a linear correlation between fluid flow velocity and depositional grain-size of the sediment. Most information about the concentration and grain-size depositional distribution in turbulence flow comes from laboratory experiments [14] and several numerical models [15]. According to those previous studies, a gradual decrease of grain-size of the sediment deposited with height above the bed surface. Similar with those previous studies, the experiment here also showed the same correlation where with the increasing height above the bed surface, there is a gradual decrease of the grain-size sediment in which deposited (figure 4).

The morphology that submarine channel initiation produce would play a role in the levee deposit and sequence composition. There in the turbulence flow mechanism would produce a fining-upward trend due to a decrease in grain-size with height. The experiment showed that coarser sediment

![Figure 2. Subdomain result showing the velocity field of the flow after the submarine channel initiation started at t=0. The geometry scale is in the meter. The rainbow scale is in standard velocity measurement (m/s)](image)
Figure 3. Cross-sectional slices of experiment result taken at 5 meters interval. Geometry scale is in the meter. The velocity field is in standard measurement (m/s).

Figure 4. Correlation between the height above the bed and grain-size sediment deposited, 
(a) curve of fluid flow velocity in X-axis against height above bed surface in Y-axis, 
(b) contoured result in map view.

at the base of the bed is deposited by the heads of the turbulence flow. This head of the turbulence flow has the highest flow activity at a lower level of the height above the bed (figure. 4a). The head of the turbulence flow has vigorous turbulent mixing and therefore transports of bigger sediment size are relatively high in the flow.
Figure 5. Cross section of geometry taken at one point event showing channel lobes at the highest activity of turbulence flow correlated with grain-size deposited; dotted shape showing bigger grain size and stripped shape showing smaller grain size.

The previous study [15] proposed depth-evolution of submarine fan channels governed by two fundamental processed, channel incision and levee building. In this experiment, the submarine channel incised by the erosion of the turbulence flow due to the magnitude of the flow. However, the experiment could not produce the result in case the fluid flow overspill that may produce another geometry of submarine fans at the time-dependent function. Another run of experiments would suffice that statement, yet still, the computational fluid dynamic simulation could not simulate the morphological result in the time-dependent function where after a series of submarine channel evolution would happen another submarine channel initiation since the setup and equation are far too complex. Hence in this experiment only able to present the geometry of the submarine channel initiation at one event only (figure 5).

Even though the experiment was done with the computational fluid dynamic simulation, there were several things that became a downside. First of all, the complexity of the turbulence flow was very hard to simulate due to locked Reynolds number that it always need 2000 or more as a nondimensional parameter. Second, the unchanged Reynolds number would result in only geometry size or inflow velocity parameter that able to be adjusted. Both of this would result in another problem, where if the geometry of the input model is too large then the computational result would not show a greater significant change in morphodynamic of submarine channel initiation. The same goes to the inflow velocity, if the velocity is too high then it basically the same as a flooding mechanism over a surface at initial time $t = 0$. Last, the COMSOL were able to compute an even bigger number of iterations, but the simulation clearly took heavy computational memory and time. Hence this experiment was only used 25 number of iterations due to the limited specification that the computer used to handle the computational matter.

Nevertheless, the simulation using COMSOL were still represented a satisfactory result, given how the idea implemented through geometrical model input and it has a feature where the desired parameters could be configured. Even though the physical experimental approach will still be needed to cross check the computational simulation results.

4. Conclusion
The experiment was done in order to simulate the turbulence flow in the submarine channel initiation process by using COMSOL Multiphysics® as the computational media for the computational fluid dynamic simulation. The result of the experiment was good enough to represent the idea of how the velocity field would be at the submarine channel initiation process and giving a good linear insight of how velocity field related to the grain-size the fluid flow would bring. There were some upside and downside things that this experiment will still need a proper physical experimental approach in laboratory scale where the surrounding area could be conditioned as nearly identical to the deep water.
parameter condition such as hydrostatic pressure, fluid-fluid interaction, fluid-solid interaction and the turbulence flow.

Acknowledgments
The major contribution came from Quantitative Sedimentology Research Group, Universitas Indonesia, financially supported by Universitas Indonesia through Hibah Publikasi Internasional untuk Tugas Akhir Mahasiswa (Hibah PITTA) under contract No. 2324/UN2/R3.1/HKP.05.00/2018.

References
[1] Piper D J W and W R Normark 2001 AAPG Bulletin 85 1407-38
[2] Middleton G V and Hampton M A 1973 Sediment gravity flows mechanics of flow and deposition Turbidities and Deep-Water Sedimentation (Los Angeles: SEPM Pacific Section) pp 1-38
[3] Posamentier H W and Kolla V 2003 J. Sediment. Res. 73 367-88
[4] Covault J A, Shelef E, Traer M, Hubbard S M, Romans B W and Fildani A 2012 J. Sediment. Res. 82 25-40
[5] Mayall M, Jones E and Casey M, Mar. Pet. Geol. 23 821-41
[6] Covault J A, Sylvester Z, Hubbard S M, Jobe Z R and Sech R 2016 The Sedimentary Record 14 4-11
[7] de Leeuw J, Eggenhuisen J T and Cartigny M J B 2016 Nat. Commun. 7 1-7
[8] Talling P J et al. 2015 Nature 450 541-4
[9] Metivier F, Lajeunesse E and Cacas M C 2005 J. Sediment. Res. 75 6-11
[10] Hoyal D C J D and Sheets B A 2009 The 33rd Int. Association of Hydraulic Research Congress, Vancouver (Canada) (Red Hook, New York: Curan Associates, Inc.)
[11] Hamilton P B, Strom K and Hoyal D C J D 2014 Sedimentology 60 1498-525
[12] Ajeunesse E, Malverti L, Lancien P, Armstrong L, Metivier F, Coleman S, Smith C E, Davies T, Cantelli A and Parker G 2010 Sedimentology 57 1-26
[13] Hamilton P B, Strom K and Hoyal D C J D 2015 J. Geophys. Res. Earth Surf. 120 1369-89
[14] Nino Y, Lopez F and Garcia M 2003 Sedimentology 50 247-63
[15] de Leeuw J, Eggenhuisen J T and Cartigny M J 2017 Sedimentology 65 931-51