The Balance-Characteristic Numerical Method on Triangle Grids.

D Y Gorbachev\textsuperscript{1}, V M Golovizin\textsuperscript{1}
\textsuperscript{1} Lomonosov Moscow State University, Faculty of Computational Mathematics and Cybernetics, Laboratory of Industrial Mathematics, Russia
E-mail: lim@cs.msu.ru

Abstract. In this paper we introduce new balance-characteristic method on triangle grids. We describe main steps of the algorithm on the two-dimensional shallow water equations. Method verified on basic test for shallow water equations with flat bottom and compared with the results of other authors.

1. Introduction
The balance-characteristic numerical method CABARET [1] shows good results in wide area of CFD problems. For example, CABARET was verified on solving simple transfer equation [2], gas dynamics problems [3] and industrial problems such as: aeroacoustics [4], shallow water over a rough bottom [5], containment modeling and many others. Despite all the advantages of this method it has restriction to use quadrangular computational grids. But in many industrial problems it is not acceptable because of many reasons: computational domain is too complex to be meshed with quadrangle, time step is too small on quadrangular grids in comparison with triangle grids with the same quality, legacy applications that are still in use accepts only triangle grids and so on.

We introduce a new balance-characteristic numerical method that aims to solve this problem without losing the properties inherent in the CABARET method.

2. Problem definition
We consider two-dimensional shallow water equations with flat bottom and subsonic flows:

\[
\begin{align*}
\frac{\partial H}{\partial t} + \frac{\partial H u_x}{\partial x} + \frac{\partial H u_y}{\partial y} &= 0 \\
\frac{\partial H u_x}{\partial t} + \frac{\partial \left( H u_x^2 + \frac{gH^2}{2} \right)}{\partial x} + \frac{\partial H u_x u_y}{\partial y} &= 0 \\
\frac{\partial H u_y}{\partial t} + \frac{\partial H u_x u_y}{\partial x} + \frac{\partial \left( H u_y^2 + \frac{gH^2}{2} \right)}{\partial y} &= 0
\end{align*}
\]

(1)

Here \( H \) - water depth, \( g \) - acceleration due to gravity, \( u_x \) and \( u_y \) velocity components in the \( x \) and \( y \) directions respectively.
3. Computational grid
Described equations are solved on an unstructured grid with triangular cells. In every triangle cell we have ten nodes: four nodes on every edge and one node at the intersection of medians. Variables related to nodes on edges and to the cell center we’re naming flux variables and conservative variables respectively. As shown on Figure 1 nodes with conservative variables will be labeled as blue circles and flux variables as black circles. Such labeling will be used in entire article. In each node we have a full set of variables: \( H \), \( u_x \) and \( u_y \).

In the context of the time dimension the template of the numerical scheme have structure as it shown on Figure 2. Like in the CABARET in variables calculation at new time step \( n + 1 \) involved variables from current time step \( n \) and half time step \( n + 1/2 \). On half step \( n + 1/2 \) only conservative variables are used.

![Figure 1. Cell template.](image1)

![Figure 2. Scheme template.](image2)

4. Numerical method
Variables calculation at the new time step is divided into three phases:
1. On the first phase conservative variables at the half time step are calculated.
2. On the second phase flux variables at the new time step are calculated.
3. Finally, On the third phase conservative variables at the new time step are calculated.

Lets describe each of three phases in details.

4.1. First phase
In this phase flux and conservative variables at the current time step are involved in calculation. First phase template to calculate conservative variables at the half time step shown on Figure 3.

![Figure 3. First phase template.](image3)

![Figure 4. Sketch of variables involved in divergence calculation for ith edge in cell.](image4)
Based on this template we can calculate new variables with the following system of equations:

\[
\begin{align*}
\frac{H_{c}^{n+1/2} - H_{c}^{n}}{\tau/2} + \frac{1}{S} \sum_{i=1}^{3} \text{integrate } (H_{i}n_{i}, n_{i}, I_{i}) &= 0 \\
\frac{(Hu_{x})_{c}^{n+1/2} - (Hu_{x})_{c}^{n}}{\tau/2} + \frac{1}{S} \sum_{i=1}^{3} \text{integrate } \left( \left( Hu_{x}^{2} + \frac{gH^{2}}{2}, Hu_{x}u_{y} \right)_{i}, n_{i}, I_{i} \right) &= 0 \\
\frac{(Hu_{y})_{c}^{n+1/2} - (Hu_{y})_{c}^{n}}{\tau/2} + \frac{1}{S} \sum_{i=1}^{3} \text{integrate } \left( (Hu_{x}u_{y}, Hu_{y}^{2} + \frac{gH^{2}}{2} \right)_{i}, n_{i}, I_{i} \right) &= 0
\end{align*}
\]

(2)

Here \( H_{c}^{n} \), \( (Hu_{x})_{c}^{n} \) and \( (Hu_{y})_{c}^{n} \) related to cell center node at \( n \) time step. \( S \) is the square of the triangle. Finally, \( \text{integrate} \) is an operation that is described as follows:

\[
\text{integrate}(n_{i}, n_{i}, I_{i}) = \left( \frac{2n_{i1} + 3n_{i2} + 3n_{i3} + n_{i4}}{8}, n_{i}, I_{i} \right)
\]

Description is done for \( i \)th edge in triangle according to the Figure 4. So \( n_{i}, I_{i} \) is \( i \)th edge normal and length respectively. Concrete values of \( n_{i} \) is taking from nodes \( i_{1}, i_{2}, i_{3} \) and \( i_{4} \). This operation calculate integral with third order of accuracy and it is also known as Simpson’s 3/8 rule.

4.2. Second phase
This phase is most important because on this phase we calculate flux variables at the new time step. This phase has a lot of variation and it is sensitive to the method used. In our method we have different procedures for calculation flux variables in edge inner nodes and triangle apex nodes (Figure 1). In this phase we operate with Riemann invariants and not with initial variables. For example, for shallow water equations for the dimension \( k \) from Figure 1 we have next Riemann invariants with corresponding velocity

\[
\begin{align*}
R &= (\pi, k) + 2\sqrt{gH} \\
Q &= (\pi, k) - 2\sqrt{gH} \\
S &= (\pi, I)
\end{align*}
\]

4.2.1. Edge inner nodes We have the following template for calculation new values in edge inner nodes:

![Figure 5. Second phase template for edge inner nodes](attachment:image.png)
For every edge inner node there is an opposite node from the same cell. Also, straight line connecting them passes through cell center node at the half time step. This facts allow us to use the same extrapolation procedure for Riemann invariants as in the CABARET.

Let’s consider node from Figure 5. For this node we calculate new values of Riemann invariants as follows:

\[ I_{n+1}^{\text{considered}} = 2I_{n+1}^{\text{center}} - I_{n}^{\text{opposite}} \]

Then we apply nonlinear correction procedure for this new value:

\[
I_{n+1}^{\text{considered}} = \begin{cases} 
    \max + \tau Q, & I_{n+1}^{\text{considered}} \geq \max + \tau Q \\
    \min + \tau Q, & I_{n+1}^{\text{considered}} \leq \min + \tau Q \\
    I_{n+1}^{\text{considered}}, & \text{otherwise}
\end{cases}
\tag{3}
\]

Here \( \max = \max\{I_{n}^{\text{considered}}, I_{n}^{\text{center}}, I_{n}^{\text{opposite}}\} \), \( \min = \min\{I_{n}^{\text{considered}}, I_{n}^{\text{center}}, I_{n}^{\text{opposite}}\} \) and \( Q \) calculates as follows:

\[
Q = \frac{I_{n+1/2}^{n} - I_{n}^{n}}{\tau/2} + \lambda_I (\text{grad} I, \overline{k})
\]

\( \text{grad} I \) calculates in the same manner as it done for divergence in first phase.

When all new values of Riemann invariants are calculated we can find initial variables. This can be done uniquely in subsonic flows. To choose concrete set of Riemann invariants to perform this operation we compare values of corresponding velocity from adjacent cells.

4.2.2. Triangle apex nodes

The calculation of new variables in triangle apex nodes occurs after calculation is performed in edge inner nodes. This allows us to use variables at the new time step \( n+1 \) from edge inner nodes.

Let’s consider some apex node with adjacent cells. We can calculate average value of \( H \) and \( \pi \) at this node by values of variables from adjacent edge inner nodes at the new time step \( n+1 \). For every Riemann invariant we can select cell from which it transfers base on these average \( H \) and \( \overline{u} \). This is done by comparison values of projected corresponding Riemann invariant velocity. Projection of the velocity is done in the directions which are constructed by connecting this apex node and center node of adjacent cells.

For example, some cell was selected for one of the Riemann invariants and we have the following template:

![Figure 6. Second phase template for triangle apex nodes](image)

Direction \( \overline{k} \) for this step is different from direction that we select for edge inner nodes. In this direction we are searching node at the intersection of line corresponding to \( \overline{k} \) and line
corresponding to characteristic curve. We reconstruct value of the Riemann invariant in this node by using interpolation polynomial in this cell. This interpolation polynomial is built on ten nodes and has third order of accuracy.

Finally, new value of the Riemann invariant we calculate as follows:

\[ I_{n+1}^{\text{considered}} = I_{n}^{\text{interpolated}} + \tau Q \]

Here \( Q \) calculates in the same way as it done for edge inner nodes.

So we can calculate new values for each Riemann invariant in considered node and reconstruct values of initial variables.

4.3. Third phase
In this phase we calculate conservative variables at new time step \( n + 1 \). In this calculation flux variables at new time step \( n + 1 \) and conservative variables at half time step \( n + 1/2 \) are involved. The system of equations for this phase can be constructed similar to the one from the first phase.

5. Water drop problem
This test is taken from the article [7] from section 4.5. For this test we have 7776 number of cells and use CFL equal to 0.3.
Our results are consistent with the results from the mentioned article with flat bottom and from the article [8].

6. Conclusions
A new balance-characteristic method has been developed and tested to simulate shallow water flows. Advantages of the present method: the capabilities of handling complex geometry by using unstructured grids, both usage conservative and characteristic representation of initial hyperbolic equations in the natural manner.

Acknowledgments
This work was supported by the Russian Science Foundation, project no. 19-11-00104.

References
[1] Goloviznin V M, Zaitsev M A, Karabasov S A, Korotkin I A 2013 Novel Algorithms of Computational Hydrodynamics for Multicore Computing (Moscow: Moscow State University Press)
[2] Goloviznin V M, Karabasov S A, Kobrinski I M 2003 Balanced-characteristic schemes with staggered conservative and transport variables Math. Model. 15 2948
[3] Goloviznin V M 2006 Balanced-characteristic method for numerical solution of one-dimensional Euler equations Math. Model. 18:11(2006) 14-30
[4] Goloviznin V M, Karabasov S A, Kozubskaya T K, Maksimov N V 2009 CABARET scheme for the numerical solution of aeroacoustics problems: Generalization to linearized one-dimensional Euler equations Comput. Math. Math. Phys. 49:12 (2009) 2168-2182
[5] Goloviznin V M, Isakov V A 2017 Balance-characteristic scheme as applied to the shallow water equations over a rough bottom Comput. Math. Math. Phys. 57:7(2017) 402-414
[6] Goloviznin V M, Chetverushkin B N 2018 New generation algorithms for computational fluid dynamics Comput. Math. Math. Phys. 58:8 (2018) 1271-1225
[7] Xing Y, Zhang X 2013 Positivity-preserving well-balanced discontinuous galerkin methods for the shallow water equations on unstructured triangular meshes Journal of Scientific Computing 57(1) 1941
[8] San O, Kara K 2011 High-order accurate spectral difference method for shallow water equations Int. J. Res. Rev. Appl. Sci. 6 41-45
[9] Yoon T H, Kang S K 2004 Finite volume model for two-dimensional shallow water flows on unstructured grids Journal of Hydraulic Engineering 130(7) 678-688
[10] Anastasiou K, Chan C T 1997 Solution of the 2D shallow water equations using the finite volume method on unstructured triangular meshes International Journal for Numerical Methods in Fluids 24(11) 1225-1245