GEOMETRIES ANALYZER

This module is used to analyze force acting on geometrical objects and export results as file in text format (comma-separated values format, .csv). Data for the analysis is taken from the results file (*.mdem), therefore saving step which was used during the simulation can influence the accuracy of the results. Linear interpolation is used to obtain data between saved time points. Also, this module can be used to analyze the agglomerate breakage, as the force acting on the agglomerate will be the same as the force acting on the corresponding geometrical object (by using 3rd Newton's law).

User can obtain information about the following parameters of geometries (*Fig. 1 - Property*):

- *Force total:* forces acting on selected geometry;
- *Distance:* Translational displacement of the wall relative to its initial position (time 0).

Geometry for the analysis can be selected from the list of available geometries (*Fig. 1 - Geometry*).

Each property can be obtained for specific time interval, divided into classes (*Fig. 1 - Time*). Values will be exported for each time point from interval [T_{FROM}; T_{TO}] starting from T_{FROM} with specified time step ΔT. For example for initial data T_{FROM} = 0 s, T_{TO} = 5 s, ΔT = 1 s, the results will be generated for time points 0, 1, 2, 3, 4, 5 s. To get values for only one specific time point T specify T_{FROM} = T_{TO} = T. There is also a possibility to obtain results only for those time points, which were saved during the simulation (thereby avoiding the interpolated results).

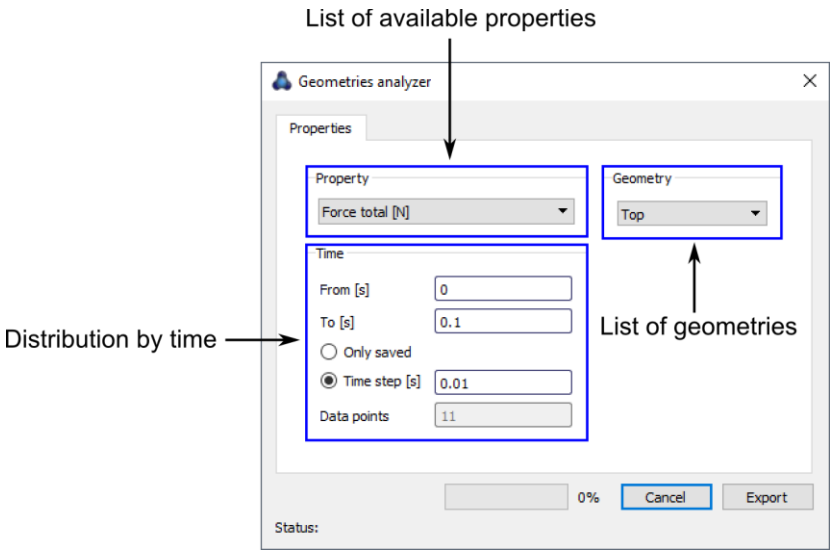

Figure 1. Main dialog of the Geometries Analyzer.