Short Communication

Numerical Simulations of Polydispersed Suspension Sedimentation in Ansys CFX

Krzysztof Kołodziejczyk*

Department of Power Engineering and Environmental Protection, Faculty of Mechanical Engineering and Robotics, AGH University of Science and Technology, Mickiewicza 30, 30-059 Kraków, Poland

Received: 11 June 2013
Accepted: 21 January 2014

Abstract

The aim of this work is to assess the possibilities of numerical simulation of the industrial slurry sedimentation process using ANSYS CFX software. Our work focused on the analysis of the simulation models available in the CFX module of the ANSYS to run the simulation of sedimentation based on the results of laboratory tests.

Keywords: sedimentation, numerical simulation of sedimentation, sedimentation test

Introduction

Sedimentation is one of the most widely used processes in water treatment systems, both in water and sewage treatment. It is a process used in both industrial and civil systems.

The popularity of this process in practical applications can be attributed to many factors. The core ones may include low operating costs of the facilities and the relatively simple structure thereof.

The essence of sedimentation consists in particles (solids) being moved down by gravity in fluid, with the simultaneous occurrence of differences of density between the falling solids and the medium in which they are located. The extensive use of sedimentation in practice has resulted in a number of mathematical and empirical models describing the process, or the effects of sedimentation equipment [1-3]. However, in most cases models describing sedimentation do not allow one to determine the concentration distribution inside a sedimentation device during analysis.

A very important change in the capacity to analyze the operation of settling devices and the process of sedimentation has been achieved by numerical methods, the use of which allows not only for obtaining information on the flow field inside the sedimentation device [4-7], but also the modeling of multiphase systems [8-12]. Moreover, the application of numerical methods to calculate the process allows the calculation of its changes over time, apart from the slurry concentration distributions in the system. Modeling of multiphase systems, which include suspensions, is still discussed [13]. Unfortunately, due to the very wide range of factors that influence the properties of suspensions [3, 14], various kinds of effects that occur during the process, depending on the concentration, as well as the materials of the dispersed fraction, there are no numerical models that would allow for the modeling of sedimentation with the same accuracy. The reasons for this lie in the level of complexity of interactions that occur during sedimentation. Depending on the physicochemical properties of the suspension (for example its concentration, granulometric composition, structure of the dispersed fraction, etc.) [14, 15] we obtain different processes for suspensions with similar parameters. An additional complicating factor in the modeling of sedimentation is the presence of areas of very high and very low concentrations within the same system [16, 17].

Such a large number of factors affecting the process makes it virtually impossible to perform a reliable numerical simulation of sedimentation without prior validation of the mathematical model used.

*e-mail: krkolodz@agh.edu.pl

Information on the distribution of slurry concentration in the settler would be a great tool in the hands of the designer, who, on this basis, could optimize the device's construction in order to obtain the highest efficiency of its operation while minimizing its size.

The object of the work described in this paper is the selection and verification of numerical models available in the commercial Ansys code CFX [18-22] used for the modeling of sedimentation of industrial suspensions. In the described research, uses the suspension derived from the process of flotation of copper ore. The suspension is characterized by a relatively high initial concentration of 130 kg/m^3 and a polydisperse granulometric composition [16].

Simulations were intended to provide an answer to whether one can obtain conformity of simulation results with laboratory tests for a suspension with known properties (known solid particle composition) at a level which would allow for the use of numerical simulations carried out in practice by the designer, or not.

Laboratory Tests

For comparison of the results obtained with the results of numerical simulations in real-life systems, it was decided to test the use of sedimentation of suspension. The test is illustrated by changing the height of the clear liquid separation zone and the suspension function of time in a measuring cylinder [3]. The primary test applied in the performed analysis was sedimentation in a measuring cylinder oriented vertically. As a control analysis, numerical simulation of sedimentation in a measuring cylinder positioned at the angle α=45° to the substrate also was carried out (the possibility to verify the process of sedimentation, and also the process of the suspension sliding down the glass wall of the settler – a phenomenon occurring in every sedimentation tank using multi-stream sedimentation).

The measurements were made in a gradated cylinder with an inner diameter of 42 mm and the suspension column height at 930 mm. The measurements used industrial slurry derived from the process of flotation of copper ore. The granulometry curve, described by the log-normal distribution density curve with parameters m=3.091 and σ=0.367, is presented in Fig. 1. Fraction-suspended solid material had the density ρ_d=2,700 kg/m^3. The initial concentration of the slurry used in the study, and then in the simulations, was s_{susp}=130 kg/m^3. This is the concentration of the suspension immediately after the process in an industrial plant. In the numerical simulations, defining the multiphase system, the volume share of each phase in the slurry is used, i.e. the volume fraction is calculated from formula (1) for the tested suspension. It will be r_d=0.04815 m^3/m^3.

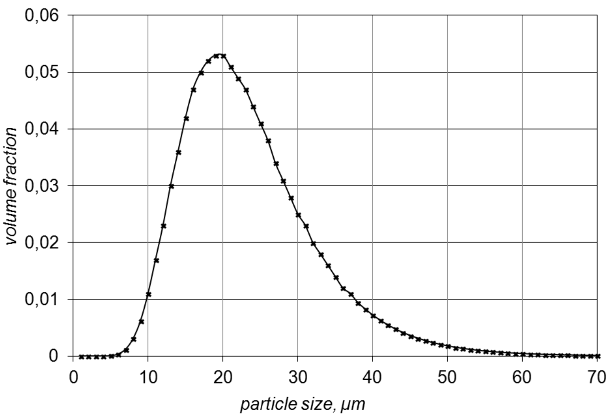

Fig. 1. Particle size distribution – log-normal curve with parameters m=3.091 and σ=0.367.

$$r_d = \frac{s_{susp}}{\rho_d} \tag{1}$$

The Numerical Model of Multiphase Systems in Ansys CFX

The Ansys CFX software package provides two classes of models of multiphase systems: Euler-Lagrange class models and Euler-Euler class models. Due to the fact that numerical simulations using the Euler-Lagrange class model are associated with running calculations for each grain of the dispersed fraction separately – depending on the computing power – it is possible to simultaneously track up to a few thousand grains. These calculations are dedicated to be carried out for low particle concentrations, such as in pollution dispersion systems. However, in systems with high concentrations, where one cubic centimeter and as many as even a few million grains could be found, the implementation of this type of calculation is not possible. In such systems, models of choice are Euler-Euler class ones, in which both the continuous fraction and the dispersed fraction are treated as the continuous phase with varying volume fraction for each of them. Total share volume for the fractions rα must be 1 equation (2) [18].

$$\sum_{\alpha=1}^{N} r_\alpha = 1 \tag{2}$$

The CFX module offers three Euler-Euler class models to calculate multiphase systems: the Particle Model, the Mixture Model, and the Free Surface Model. The model dedicated for calculation of the disperse phase in the form of grains is the Particle Model used in the analysis.

The continuity equation for one faction, assuming no mass transfer between phases and the absence of mass sources, will be described by equation (3) [18]

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha \mathbf{U}_\alpha) = 0 \tag{3}$$

In addition to the incompressible fluid, the continuity equation will have the following form (4) [18].

$$\sum_\alpha \nabla \cdot (r_\alpha \mathbf{U}_\alpha) = 0 \tag{4}$$

The momentum equation for the fluid phase, on the other hand, describes relationship (5) [18]:

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha \mathbf{U}_\alpha\right)+\nabla\cdot\left(r_\alpha(\rho_\alpha \mathbf{U}_\alpha \otimes \mathbf{U}_\alpha)\right)= -r_\alpha \nabla p_\alpha + \nabla\cdot\left(r_\alpha \mu_\alpha (\nabla \mathbf{U}_\alpha + (\nabla \mathbf{U}_\alpha)^T)\right)+\mathbf{S}_{M\alpha}+\mathbf{M}_\alpha \quad (5)$$

...where $S_{M\alpha}$ describes momentum sources due to external body forces and M_α describes the interfacial forces acting on phase α due to the presence of other phases.

In the calculations, the influence of the model describing the drag coefficient (interphase drag) in the Particle Model for the process of sedimentation was analyzed.

The interphase drag is defined by equation (6):

$$\mathbf{M}_\alpha = c_{\alpha\beta}(\mathbf{U}_\beta - \mathbf{U}_\alpha) \quad (6)$$

At the same time, the coefficient $c_{\alpha\beta}$ is described by equation (7):

$$c_{\alpha\beta} = \frac{C_D}{8} A_{\alpha\beta} \rho_\alpha \left|\mathbf{U}_\beta - \mathbf{U}_\alpha\right| \quad (7)$$

...where the drag C_D can be calculated by the application of one of the equations:

Schiller Naumann Drag Model (8) [21] – dedicated for calculations of multiphase systems with a small share of the disperse phase.

$$C_D = \begin{cases} \frac{24}{\mathrm{Re}}(1+0.15\mathrm{Re}^{0.687}), & \mathrm{Re} \le 1000, \\ 0.44, & \mathrm{Re} > 1000, \end{cases} \quad (8)$$

Wen Yu Drag Model (9) [22].

$$C_D = r_c^{-1.65} \max\left(\frac{24}{\mathrm{Re'}}(1+0.15\mathrm{Re'}^{0.687}), 0.44\right) \quad \mathrm{Re'} = r_c \mathrm{Re} \quad (9)$$

...where r_c – continuous phase volume fraction.

Gidaspow Drag Model (10) [19].

$$C_D = C_D(Wen\ Yu), \quad r_c > 0.8$$

$$c_{\alpha\beta} = 150\frac{(1-r_c)^2 \mu_c}{r_c d_p^2} + \frac{7}{4}\frac{(1-r_c)\rho_c \left|\mathbf{U}_c - \mathbf{U}_d\right|}{d_p}, \quad r_c < 0.8 \quad (10)$$

Numerical Simulations of Sedimentation

In the analysis conducted, several numerical simulations of suspended particle sedimentation were performed, with different models and different parameters of the modeled suspension assumed, in order to obtain information about the impact of different models and their parameters on the process of sedimentation as well as the achieved convergence with the results of calculations carried out during laboratory tests.

All numerical simulations were performed using a computational Particle Model belonging to the class of Euler-Euler models. Numerical simulations were carried out assuming a cylinder with geometrical parameters identical to the one tested. The cylinder was divided by a hexahedral grid consisting of 27,561 elements (Fig. 2). The simulations were implemented as variables in time, with the time step of 1 s. The time step was chosen on the basis of prior conducted analyses [10, 11]. In view of the fact that the sedimentation test is implemented as a static test (no flow) the boundary condition for all the walls of the geometry, the assigned parameter was "wall." In each simulation run, as the initial condition of even concentration of suspended slurry throughout the volume was set, determined by the use of appropriate share volumes for each phase and modeled fraction of the suspension. The conducted simulations also assumed a maximum compression rate of the dispersal phase at 0.222, which corresponds to the maximum concentration in the layer of compression amounting to 600 kg/m^3.

Numerical Model of Polydispersed Suspension

In the analysis, polydisperse suspension was used with the density function of the particle size distribution as shown in Fig. 1. The numerical simulations carried out using Euler-Euler class models do not make it possible to define a function of particle size of the dispersal phase. In practice, the polydisperse size distribution is carried out through several phases of dispersed particles of different diameters and fraction volumes. For the slurry used in the study, two grain compositions were prepared, with 10 and 5 grain fractions, respectively. On the basis of the particle size distribution (Fig. 1), for each fraction the share volume of grains found in the range between the lower and upper limit was determined. For each interval, also a substitute grain diameter was determined within the range, with the value specified for approximately d_{50} grain of the range. Table 1 presents data on the granulometry with the suspension divided into 10 grain fractions, whereas Table 2 shows the suspension divided into five grain fractions.

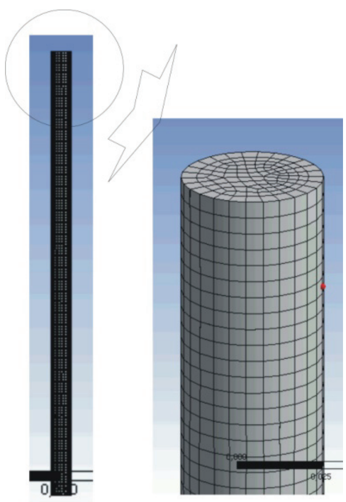

Fig. 2. The geometry used in the simulations divided into a grid of elements.

Table 1. Granulometry – 10 fractions of the dispersed phase.

No.	Particle diameter	Volume share	Interval limits	
			Low	High
	[μm]	[-]	[μm]	[μm]
1	5	0.00008	0	7.5
2	10	0.00289	7.5	12.5
3	15	0.00986	12.5	17.5
4	20	0.01242	17.5	22.5
5	25	0.00982	22.5	27.5
6	30	0.00615	27.5	32.5
7	35	0.00341	32.5	37.5
8	40	0.00177	37.5	42.5
9	50	0.00154	42.5	57.5
10	70	0.00021	57.5	∞

Table 2. Granulometry – 5 fractions of the dispersed phase.

No.	Particle diameter	Volume share	Interval limits	
			Low	High
	[μm]	[-]	[μm]	[μm]
1	13	0.00714	0	15
2	20	0.02349	15	25
3	30	0.01256	25	35
4	40	0.00435	35	50
5	60	0.00061	50	∞

Simulations for 5- and 10-Fractions of the Dispersed Phase

The first simulation was performed to calculate a suspension, whose granulometry has been described by the histogram for five and ten fractions of the dispersal phase, the granulometric compositions for which are listed in Tables 1 and 2. The analyses carried out so far [10, 11] have shown that process modeling suspensions of polydisperse grain composition using a monodisperse suspension model do not provide positive results. Therefore, an attempt to verify the sensitivity of numerical simulations was carried out to describe the particle size distribution of the suspension. The calculations were made assuming a laminar flow model for the continuous phase, the calculation taking into account interphase drag using the Gidaspow model [19]. The results of the numerical simulation paired with laboratory test results are found in the graph (Fig. 3).

As can be seen in the chart, the reproduced sedimentation curve for the suspension modeled using five grain fractions coincides with the course of the curve for the suspension of 10 granulometric fractions. At the same time, the shape of the simulation curves, despite their initial compatibility with the shape of the curve from the laboratory tests, was finally significantly different. Based on these simulations, we can conclude that the test suspension does not produce significant differences in the simulation using the 5 and 10 dispersed fractions. In the following simulations, the dispersed phase of the suspension was defined using a model of suspension of five granulometric factions.

Simulations Involving Drag Models

The second analysis was conducted to determine which of the models in available ANSYS CFX, used for modeling interphase drag, will best fit the curve of sedimentation resulting from the simulation, to laboratory test results. Moreover, the conducted simulations should provide an answer to what extent the use of different models of interfacial drag impacts the process of a test suspension's sedimentation.

The calculations were performed with the use of the following drag models: Schiller Naumann (8), Wen Yu (9),

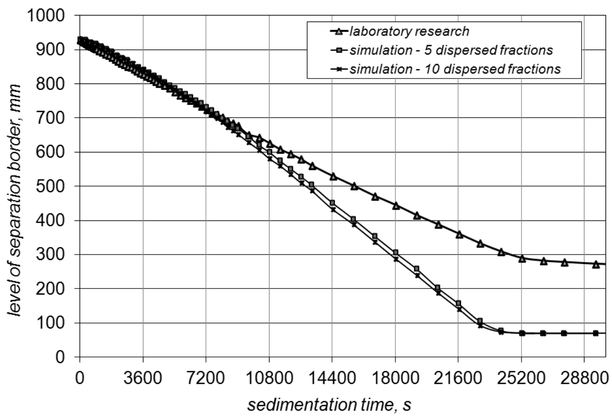

Fig. 3. Sedimentation curve obtained in laboratory studies and numerical simulations for the five and ten grain fractions of the dispersed phase of suspension, respectively.

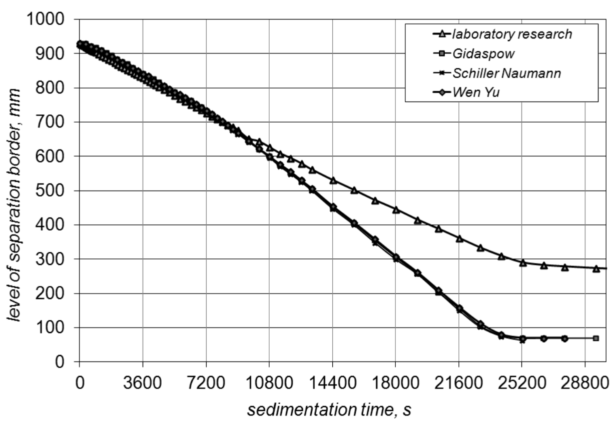

Fig. 4. Sedimentation curve obtained in laboratory studies and numerical simulations for the three models of interphase drag: Schiller Naumann, Wen Yu, and Gidaspow.

and Gidaspow (10). The result of the conducted simulations, expressed as sedimentation curves, is presented in the graph (Fig. 4). Calculations are performed (similarly as in the first case) by applying a laminar flow model for the continuous phase, with the dispersal phase consisting of 5 granulometric fractions.

The obtained results clearly show that regardless of the drag model of choice the sedimentation process is the same – the curves obtained in the numerical simulation coincide, in the absence of compliance with the laboratory results.

Simulations Involving Turbulence and Solid Particle Collision Models

The next step of the analyses was to find the parameters for analysis that would allow for a higher level of compliance with the sedimentation test simulations and laboratory tests. In addition to the Gidaspow drag model, the calculation also takes into account the interaction between the disperse phase through the use of the Solid Particle Collision Models [18], which are based on the kinetic theory of gases, taking into account inelastic collisions. The calculations were run with the application of the laminar model of the continuous phase flow, applied on the basis of the rate of 0.9, which was derived from analysis of the literature. The results are presented in Fig. 5, where the reference also contains the curve from the laboratory tests, as well as a curve that does not consider particle collision models. Fig. 5 also includes a sedimentation curve obtained for the turbulent flow model of the continuous phase, obtained using the turbulence model k-ε.

As one can see from the presented results, the application of particle collision models significantly alters the rate of sedimentation, but not in the expected direction. An even greater difference in the process is obtained for the calculation of flow using the turbulent computational model. In both cases we observe a significant increase in the rate of sedimentation.

Analyzing the results of the process from all the simulations conducted, we see that, in any case, regardless of the computational model, the separation border between the layer of sludge and pure liquid is found around 50-70 mm from the bottom of the container, whereas in the case of sedimentation carried out in the laboratory the level obtained was 280 mm. Given the initial concentration of the suspension, which was 130 kg/m^3, and the initial height of the suspension layer of 930 mm, one can calculate the average concentration in the layer of compression, which in laboratory testing will be about 430 kg/m^3, and in numerical simulations will be approximately 2,160-1,730 kg/m^3. This value is significantly different from the compression level achieved in laboratory testing, but also much higher than assumed in the study of the maximum compression level amounting to 600 kg/m^3. The resulting mismatch is confirmed in the literature [18, 20] and may be one of the elements explaining the variance in the results of laboratory and simulations.

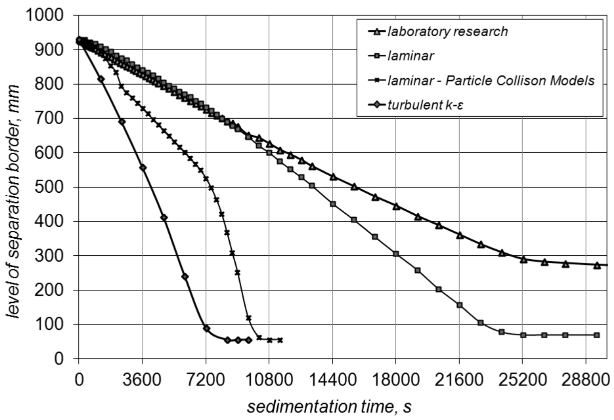

Fig. 5. Sedimentation curves obtained in laboratory studies and numerical simulations of laminar and turbulent flow model of the continuous phase.

Conclusions

Our paper presents the results of numerical simulations of sedimentation processes carried out in the Ansys module CFX. The analyses were conducted only on the basis of the known properties of the suspension (particle size distribution, density, and sedimentation characteristics such as maximum compression).

The analysis shows that, for the slurry used in the simulations, the resulting curve describing the process of sedimentation does not contain significant differences in the modeling of the dispersed phase using, respectively, five and ten granulometric fractions. For the modeled suspension, five grain fractions appear to be sufficient to carry out the simulation.

The conducted analysis clearly shows that while the analyzed models do allow for the modeling of multiphase systems with good accuracy for low concentration, they do not offer acceptable accuracy for modeling sedimentation in turbulent range and at very high concentrations.

Acknowledgements

The numerical calculations were carried out in ACK CYFRONET AGH as a part of the grant MNiSW/Zeus_lokalnie/AGH/030/2012.

References

1. BUSTOS M.C., CONCHA F., BÜRGER R., TORY E.M. Sedimentation and Thickening: Phenomenological Foundation and Mathematical Theory, Springer, **1999**.
2. KOWALSKI W.P. Lamella sedimentation – The state of art and directions of development. Chemical and Process Engineering **25**, (3), 1163, **2004**.
3. TORY E.M. Sedimentation of small particles in a viscous fluid. Computational Mechanics Publications, Boston, 281, **1996**.
4. BAJCAR T., GOSAR L., SIROK B., STEINMAN F., RAK G. Influence of flow field on sedimentation efficiency in a circular settling tank with peripheral inflow and central effluent. Chem. Eng. Process., **49**, (5), 514, **2010**.

5. GOULA A.M., KOSTOGLOU M., KARAPANTSIOS T.D., ZOUBOULIS A.I. A CFD methodology for the design of sedimentation tanks in potable water treatment. Chem. Eng. J. **111**, 105, **2008**.
6. KOŁODZIEJCZYK K., BANAŚ M., WARZECHA P. Flow modeling in a laboratory settling tank with optional counter-current or cross-current lamella. Journal of Achievements in Materials and Manufacturing Engineering, **53**, (1), 28, **2012**.
7. FATEMEH ROSTAMI, MAHDI SHAHROKHI, MD AZLIN MD SAID, ROZI ABDULLAH, SYAFALNI. Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks. Appl. Math. Model., **35**, (6), 3012, **2011**.
8. BARGIEL M., TORY E.M. Simulation of sedimentation and fluidization of polydisperse suspensions via a Markov model. Chem. Eng. J. **61**, 5575, **2006**.
9. GICALA B. Numerical modeling multiphase flow with disperssion phase. Theoretical grounds and application. Institute of Mining Technology KOMAG. Gliwice **2011**.
10. KOLODZIEJCZYK K. Modeling of sedimentation in lamella settling tanks with the application of numerical methods. Pol. J. Environ. Stud., **21**, (5A), 190, **2012**.
11. KOLODZIEJCZYK K. The modelling of laboratory multiflux settling tank work. Pol. J. Environ. Stud., **20**, (4A), 130, **2011**.
12. TRYGGAVASON G., ZALESKI S. Direct numerical simulations of gas-liquid multiphase flows. Cambridge Univ Pr. **2011**.
13. PROSPERETTI A., TRYGGVASON G. Computational Method for Multiphase Flow. Cambridge University Press, New York, **2007**.
14. KOWALSKI W.P. Investigation of fine grains distribution using the sedimentation analysis, Journal Of Materials Processing Technology, **157**, 561, **2004**.
15. BANAŚ M. The dependence of sedimentation efficiency on suspension concentration. Inż. Chem. i Proc. **25**, 663, **2004**.
16. KOLODZIEJCZYK K., ZACHARZ T. The suspension purification in the combined co-current and counter-current sedimentation process – Empirical model. Chemical and Process Engineering **25**, (3), 1107, **2004**.
17. KOWALSKI W.P., KOLODZIEJCZYK K., ZACHARZ T. The investigations of a lamella concurrent sedimentation process. Chemical and Process Engineering **22**, (3C), 777, **2001**.
18. ANSYS® Inc., 2010 – Academic Research, Release 13.0, Help System, CFX - Solver Theory Guide. **2010**.
19. GIDASPOW D. Multiphase Flow and Fluidisation, Academic Press, **1994**.
20. LUNDBERG J., HALVORSEN B.M. A review of some exsisting drag models describing the interaction between phases in a bubbling fluidized bed. Proc. 49th Scand. Conf. Simulation and Modeling, Oslo University College, Oslo, Norway. **2008**.
21. SCHILLER L., NAUMANN A. A drag coefficient correlation. VDI Zeits, **77**, 318, **1933**.
22. WEN C.Y., YU Y.H. Mechanics of Fluidization. Chem. Eng. Prog. Symp. Ser. **62**, 100, **1966**.