We are IntechOpen,
the world's leading publisher of
Open Access books
Built by scientists, for scientists

4,800
Open access books available

122,000
International authors and editors

135M
Downloads

Our authors are among the

154
Countries delivered to

TOP 1%
most cited scientists

12.2%
Contributors from top 500 universities

WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?
Contact book.department@intechopen.com

Numbers displayed above are based on latest data collected.
For more information visit www.intechopen.com

Integrated Technology for CAD Modeling and CAE Analysis of a Basic Hydraulic Cylinder

Radostina Petrova[1] and Sotir Chernev[2]
[1]Technical University of Sofia, Faculty of Engineering and Education
[2]"HES" PLC, Yambol
Bulgaria

1. Introduction

Usually solution of different engineering problems requires design of various objects or systems. Basically, there are three general approaches to solving engineering problems: an experimental approach, a computational approach and a computational-experimental approach, which combines both of the formentioned. Each of the first two approaches has advantages and disadvantages, while the last one joins the advantages and avoids the disadvantages of the other two. Complex engineering problems, including the presented one, are solved mainly in this way.

When selecting the most suitable computational code for solving a problem, it is obligatory to mind that each computational code is based on a mathematical model of the governing physical processes, expressed in the form of a set of equations derived from physical laws, including semi-empirical and empirical constants or relationships. Consequently, an appropriate method for solving these equations is also required.

For problems which solution is based on Finite Volume Method (FVM) (Versteeg & Malalasekera, 2007; Wendt & Anderson, 2009) the equations of the mathematical model are solved in a discrete form on a computational mesh. The solution of the mathematical problem is obtained with a certain degree of accuracy, depending on the method of discretising the differential and/or integral equations and on the method of solving the obtained discrete equations. Of course, the solution also depends on the introduced initial data. It is known that higher accurate solution requires finer computational mesh, provided through rather substantial computer memory and CPU time.

2. Theoretical background of the problem

The authors of this chapter provide and discuss an example of modeling and simulation of an engineering problem realized through SolidWorks+SW Flow Simulation+SW Motion software correctly and adequately from a physical viewpoint. The solution of this design challenge is a result of the cooperation between industry and science. Coperating company "HES" PLC, Yambol is among the best producers of hydraulic cylinders in Bulgaria. Most of its products are produced in limited series and are result of an individual design work. The idea of the study is to develop user-frendly and adaptable applicable environment, suitable

for investigation on different firm designer products. It has been decided that the study should contain the following modules:

- *Technological design of a hydraulic cylinder, specified by its structural scheme.* This scheme is a part of the production plan of "HES" PLC, Yambol and describes manufacturing of each of the cylinder's components. The module is implemented through 3D CAD system – in our case SolidWorks (SW) (Lombard, 2011).
- *Numerical simulation investigating the fluid flow inside the hydraulic cylinder.* The investigation aims to calculate the active force that pushes the Piston Rod Kit (PRK) precisely (see fig. 1 and 2), to visualize the flow trajectories and consequently, to outline the vulnerable domains in cylinder structure. This module uses SolidWorks Flow Simulation (SW Flow …, 2011; Solving …, 2011).
- *Dynamic numerical simulation on Piston Rod Kit motion.* The aim is to calculate the friction force between the Double Acting Seal (DAS) and Cylinder Tube (CT) (see fig. 1), which influences strongly on maintenance, reliability and life of parts. This module is realized through SolidWorks Motion (SW Motion) software package (SW Motion ..., 2011).

2.1 Governing equations in investigating the flow work, (SW Flow..2011; Solving..,2011)

The Navier-Stokes equations, which formulate mass, momentum and energy conservation laws for fluid flows are used. They are supplemented by fluid state equations defining the nature of the fluid, including its density, viscosity and thermal conductivity of temperature. The particular problem is specified by the necessity of defining flow's geometry, boundary and initial conditions. The authors rely entirely on software capacity of predicting laminar and turbulent flows.

Generally, the state equation of a fluid has the following form:

$$P=f(p,T,y) \tag{1}$$

where $y =(y_1, \ldots y_M)$ is the concentration vector of the fluid mixture components. In our case the gas is treated as an ideal one. The mass transfer is calculated under the following specific equation:

$$\frac{\partial \rho y_m}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i y_m\right) = \frac{\partial}{\partial x_i}\left(\left(D_{mn} + D_{mn}^t\right)\frac{\partial y_n}{\partial x_i}\right) + S_m, \tag{2}$$
$$m = 1,2,\ldots M$$

wherein D_{mn} and D_{mn}^t are molecular and turbulent matrices of diffusion and S_m is the rate of production or consumption of the *m-th* component.

In simulating the flow's motion we use calculating of local mean age (LMA). This is the average time τ for fluid to travel from the selected inlet opening to the pointed outlet, considering both the velocity and diffusion. It is determined by solving the following equation:

$$\sum_{i=1}^{3}\frac{\partial}{\partial x_i}\left(\rho \tau u_i - \left(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t}\right)\frac{\partial \tau}{\partial x_i}\right) = \rho, \tag{3}$$

where x_i is the *i-th* coordinate, ρ is the density, u_i is the *i-th* velocity component, μ is the dynamic viscosity coefficient, μ_t is the turbulent eddy viscosity coefficient, σ and σ_t are the laminar and turbulent Schmidt numbers. The equation is solved under the $\tau = 0$ boundary condition on the inlet opening. Dimensionless LMA is chosen in the provided example. This is LMA divided by the *V/Q* ratio, where *V* is the volume of the computational fluid domain and *Q* is the volume flow rate of the fluid entering the fluid volume.

The employed in this example numerical solution technique is standart, robust and reliable. Hence, it does not require any user knowledge about the computational mesh and the numerical methods employed. The used software package solves the governing equations through the finite volume (FV) method on a spatially rectangular computational mesh, designed in Cartesian coordinate system with the planes orthogonal to its axes and refined locally at the solid/fluid interface and, if necessary, additionally in specified fluid regions, at the solid/solid surfaces, and in the fluid region during calculation. Values of all the physical variables are stored at the mesh cell centers. Due to the FV method, the governing equations are discretized in a conservative form. The spatial derivatives are approximated with implicit difference operators of second-order accuracy. The time derivatives are approximated with an implicit first-order Euler scheme. The viscosity of the numerical scheme with respect to the fluid viscosity is negligible.

2.2 Theoretical background of dynamic simulation of RPK motion, (SW Motion.., 2011)

Static studies assume that loads are constant or applied very slowly until they reach their full values. Thus, the velocity and acceleration of each particle of the model are assumed to be zero. As a result, static studies neglect inertial and damping forces. For many practical cases, loads are not applied slowly or they change in time. Generally, if the frequency of a load is larger than 1/3 of the lowest (fundamental) modal frequency, a dynamic study should be used. Objectives of the dynamic analysis include: design of structural and mechanical systems that ought to operate without failure in dynamic environments and modifying system's characteristics (i.e. geometry, damping mechanisms, material properties, etc.) in order to reduce vibration effects. A dynamic simulation is also known as a kinetic simulation.

Many of the engineering products contain moving assemblies of components. For their analysis and correct design, it is necessary to perform a dynamic simulation of the mechanism. This is a time-history solution of all displacements, velocities, accelerations and internal reaction forces in the model driven by a set of external forces and excitations. Unlike kinematic and static simulations which involve the solution of only algebraic equations, dynamic simulations are more complex because they involve the solution of differential and algebraic equations (DAEs). It is the most complex and computationally demanding type of simulation and is meant to be used with models that have one or more degrees of freedom.

The basic algorithm available in SW Motion solver performs the numerical integration required for dynamic analyses based on stiff solution methods that use implicit, backward difference formulations (BDF) to solve the DAEs. It sets coupled differential and algebraic equations to define the functions of motion of the model. A numerical solution to these equations is obtained by integrating the differential equations while satisfying algebraic constraint equations at each time step. The set of differential equations is numerically stiff

when there is a wide interval between high and low frequency eigenvalues. The possible solution of the equations of motion depends on the numerical stiffness of the equations - the stiffer the equations are, the slower the solution is. Numerically stiff differential equations require stiff integration methods to compute the solution's efficiency because other types of methods for solving differential equations perform poorly and are too slow.

There are three stiff integration methods for computing motion by SW Motion solver:

- Gear (GSTIFF)
- Modified Gear (WSTIFF)
- Stabilized Index-2 method (SI2_GSTIFF), which is a modification of the GSTIFF method.

GSTIFF integration method, developed by C. W. Gear (Gear, 1971a; 1971b), is the most widely-used and tested integrator. It is a variable-order, variable-step and multi-step integrator with a maximum integration order of six. Among benefits of GSTIFF are high speed, high accuracy of the system displacements and robust in handling a variety of analysis problems. A limitation of the procedure is that velocities and especially accelerations can have errors. An easy way to minimize these errors is to control the maximum time step that the integrator is allowed to take, so that the integrator runs at a constant step size and runs consistently at a high order (three or more). Additionally, corrector failures can be encountered at small step sizes. These failures occur because the Jacobian matrix is a function of the inverse of the step size and becomes ill-conditioned at small steps. However, GSTIFF method is a fast and accurate method for computing displacements for a wide range of motion analysis problems. It ensures that the solution satisfies all constraints, although it does not ensure that the velocities and accelerations calculated satisfy all first- and second-time derivatives. The solver monitors an integration error only in system displacements, not in velocities.

WSTIFF (Brenan, 1996) is another variable order, variable step size stiff integrator. GSTIFF and WSTIFF are similar in formulation and behavior. Both use BDF. They differ in that GSTIFF coefficients are mostly calculated assuming a constant step size, whereas WSTIFF coefficients are a function of the step size. If the step size changes suddenly during integration, GSTIFF introduces a small error, while WSTIFF can handle step size changes without loss of accuracy. Sudden step size changes occur whenever there are discontinuous forces, discontinuous motions etc.

SI2_GSTIFF, a Stabilized Index-2 method, is a modification of the GSTIFF method. This integration method provides better error control over the velocity and acceleration terms in the equations of motion. Provided the motion is sufficiently smooth, SI2_GSTIFF velocity and acceleration results are more accurate than those computed with GSTIFF or WSTIFF and that is true even for motions with high frequency oscillations. SI2_GSTIFF is also more accurate with smaller step sizes, but is still significantly slower.

All of the three integrators (GSTIFF, WSTIFF, and SI2_GSTIFF) use Newton-Raphson iterations to solve the DAEs of motion. This iteration process is referred to as correcting the solution. The adaptivity value modifies the corrector error tolerance to include a term that is inversely proportional to the integration step size. This is intended to loosen the corrector tolerance when the step size gets small. If the integration step size is equal to *h*, *Adaptivity/h*

is added to the corrector tolerance. The *Adaptivity* value affects the GSTIFF, WSTIFF and SI2_GSTIFF integrators.

The control over the convergence of the calculations can be done by adjusting the values of several dynamic parametres, such as: accuracy, number of iterations, maximum step size, recalculation the Jacobian matrix.

3. Basic description of the used software technics

3.1 Detailed description of the used by Flow Simulation technics, (SW Flow..2011; Solving..,2011)

Since Flow Simulation is based on solving time-dependent Navier-Stokes equations, steady-state problems are solved through a steady-state approach. To obtain the steady-state solution quicklier a method of local (over the computational domain) time steps is employed. A multigrid method is used for accelerating the solution convergence and suppressing parasitic oscillations. The computational mesh is built by dividing the computational domain into parallelepiped cells whose sides are orthogonal to the Global coordinate system axes. Procedures of the computational mesh refinement are used to better resolve the model features, such as high-curvature surfaces in contact with fluid, narrow flow passages (gaps) and the specified insulators' boundaries. During the subsequent calculations while solving the problem the computational mesh can be refined additionally to better resolve the high-gradient flow and solid regions revealed in the calculations.

Most of the fluid flows encountered in engineering practice are turbulent, so Flow Simulation was mainly developed to simulate and study turbulent flows. To predict turbulent flows, the Favre-averaged Navier-Stokes equations are used where time-averaged effects of the flow turbulence on the flow parameters are considered. Through this procedure the Reynolds stresses appear in the equations for which additional information must be provided. To close the system of equations, Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate. Flow Simulation employs one system of equations to describe both laminar and turbulent flows. Moreover, transition from a laminar state to turbulent one and vice versa is possible. Flows in models with moving walls are computed by specifying the corresponding boundary conditions. Thus, for choosing model's characteristics it is necessary to remember how important the right choice of boundary conditions is. For internal flows, i.e., flows inside models, Flow Simulation offers the following two options of specifying the flow boundary conditions: manually at the model inlets and outlets (i.e. model openings), or to specify them by transferring the results obtained in another Flow Simulation calculation in the same coordinate system. With the first option, all the model openings are classified into "pressure" openings, "flow" openings and "fans", depending on the flow boundary conditions, which are intended to be specified on them. A "flow" opening boundary condition is imposed when dynamic flow properties (i.e., the flow direction and mass, volume flow rate or velocity) are known at the opening. The pressure at the opening is determined as a part of the solution. In Flow Simulation the default velocity boundary condition at solid walls corresponds to the no-slip condition and the solid walls are also considered to be impermeable. In addition to this, the wall surface's translation and/or rotation can be specified.

3.2 Detailed description of the used by SW Motion integration technics, (SW Motion.., 2011)

SW Motion uses complete kinematic modeling to compute component motion and dynamic calculations to analyze forces, excited in mechanisms by springs, dampers, motors, gravity, friction, etc. The large and diverse number of kinematic connections in the program's libraries enables the creation of mechanisms with varying degrees of complexity. They can be designed through setting one or more drives or through defining a trajectory or a set of trajectories of a particular point from the mechanism's parts. Tracking the motion parameters (linear and angular speeds and accelerations, etc.) as a function of time or of another chosen by the user parameter is also enabled.

For obtaining the final results SW Motion solver provides three types of integration methods: GSTIFF (default), SI2_GSTIFF and WSTIFF.

The user directly controls the following integration options:

- *Maximum Iterations* - Default value is 25. It specifies the maximum number of times the numeric integrator iterates in the search of a solution for a given time step. If the program exceeds this limit, a convergence failure is recorded.
- *Initial Integrator Step Size* – The command enters the first integration step size used by the variable step integrator. The initial integrator step size controls the speed at which the integration method starts and its initial accuracy. The user can run the simulation quicklier in subsequent runs by increasing this value.
- *Minimum Integrator Step Size* - This is the value of the lower bound of the integration time step. The simulation time can be decreased by increasing this value.
- *Maximum Integrator Step Size* – This enters the upper bound of the integration time step. This is important if the integration method does not detect short-lived events such as impacts. Otherwise, it is recommended this value be of the same order as the short-lived events. If the user sets this value too large, some events can be ignored by the simulation.
- *Jacobian Re-evaluation* – It enables the frequency of matrix re-evaluation to be controlled. More frequent re-evaluation gives better simulation accuracy at the cost of simulation time. If the model does not change significantly over time, a smaller Jacobian re-evaluation option can be used.

3.3 Friction phenomenon in SW Motion, (SW Motion…, 2011)

Friction is a resistive force that occurs in joints and between parts in contact. When parts are in contact, friction is calculated based on the static and dynamic coefficients of friction and the normal force acting on the part. The static and dynamic friction properties applied to the contact calculation are basically derived from the material properties. SW Motion incorporates dynamic friction into the contact calculation. Contact friction is the friction that occurs between bodies in contact. The following friction parametres can be monitored:

- *Dynamic Friction Velocity* - Specifies the velocity at which dynamic friction becomes constant.
- *Dynamic Friction Coefficient* - Specifies the constant used to calculate forces due to dynamic friction.
- *Static Friction* - Includes static friction in the contact calculation.

- *Static Friction Velocity* - Specifies the velocity at which the static frictional force is overcomed so that a stationary component begins to move.
- *Static Friction Coefficient* - Specifies the constant used to calculate the force necessary to overcome forces between two touching bodies at rest.

In the presented example the contact friction force is calculated. The velocities and coefficients of friction used are assigned automatically based on the predefined for each contact set materials. However, they may not be the most appropriate parameters based on the dynamics of the model. Then, if necessary, these coefficients can be set manually. SW Motion uses the Coulomb friction method and fits a smooth curve to the friction parameters to solve the friction force.

4. Basic description of the used SolidWorks software

4.1 CAD/CAE systems

The integration of CAD/CAE systems in developed manufacture gives experts the opportunity to put fewer resources and less energy into technical activities supporting the process of design. Thus they focus their effort and intellectual capability on creating innovative and optimal solutions or on generating new ideas for solving known or new problems and challenges. CAD (Computer Aided Design) by definition means a combination of hardware and software used for optimal solution of the geometric problems in product design. CAD systems solve mainly tasks related to describing the geometry of components, the assemblies and products as a whole. In recent years the capabilities of CAD systems have been expanded. Today they have been successfully integrated to other subsystems (CAE, CAM, etc.), aiming to avoid manual reformulation of data and to connect individual work environments. While the term CAD includes all the geometry-oriented tasks, CAE (Computer Aided Engineering) covers all the computing tasks that take place within the designing process. This involves both all calculations during the designing and all optimising procedures in achievement of constructive solutions. There is a steady interaction between CAD and CAE, since the geometry generated by the CAD system is often the basis for CAE and vice versa (Янакиев & Николов, 2010; Топалова & Бакърджиев, 2006).

The creation and processing of the product's digital model is the basis of the engineering design methodology using CAD/CAE systems. It includes geometric characteristics, data about the material and the properties of the product, some additional specific information for the designed product whereas the main component is the geometrical model. It is created and displayed as a graphic image on the screen display. The principles of operation in geometric modeling are independant from the software environment. The impact of CAE technologies on engineering has been increasing since the second half of the previous century so that it could encounter the enforced requirements for higher precision and suitability of the model and environment. Modern development of computing devices and software allows the users to model the process of deformation of complex-shaped components and structures made of different materials.

4.2 SolidWorks (SW) CAD environment

SolidWorks is also used as a software platform for many other programs. SolidWorks is an integrated CAD/CAE system, providing a unified and interconnected environment for

design, for structural and dynamic analyses, for data management of engineered products. It was developed in 1993 and since 1997 has been a trade brand of French company Dassault Systèmes. SolidWorks is known to be among the most widely used standard softwares for 3D designing. The system incorporates modules for preparing technical documentation, kinematic, dynamic, thermal and strength analysis of structures, flow simulation technics, design of specific product (sheet metal products, mold products, etc.) and so on.

Based on a multifunctional interface, SW has a parametric, 100% associative and hybrid modeler. The large software libraries provide a lot of parametric 3D parts, ready to be implemented in the model. SolidWorks enables joining of various parts together and developing different structures and mechanisms. It enables establishement of complex mechanical relations, checking the assemblies for interference, collisions and alignment of the parts. SolidWorks imports/exports files to AutoCAD, DraftSight (a free product developed by Dassault Systèmes) and some other well-known software products.

4.3 CAD modeling of a hydraulic cylinder in SW environment

For the establisment of a CAD model, design drawings of all parts, constituting the hydraulic cylinder as a part of the production program of "HES" PLC, Yambol are used. The developed hydraulic cylinder is a double acting cylinder of a piston type. Both piston courses (forward and backward) are carried out under the influence of a working fluid (hydraulic oil). The components of the hydraulic cylinder are shown in fig.1.

1. Piston Rod (PR)
2. Cylinder Front Head (CFH)
3. Nut Port (NP)
4. Cylinder Body/Tube (CT)
5. Double Acting Seal (DAS)
6. Piston (Pst)
7. Cap End Head (CEH)
8. Plate Bearing (PB)
9. Wiper (Wpr)
10. Rod Seal (RS)
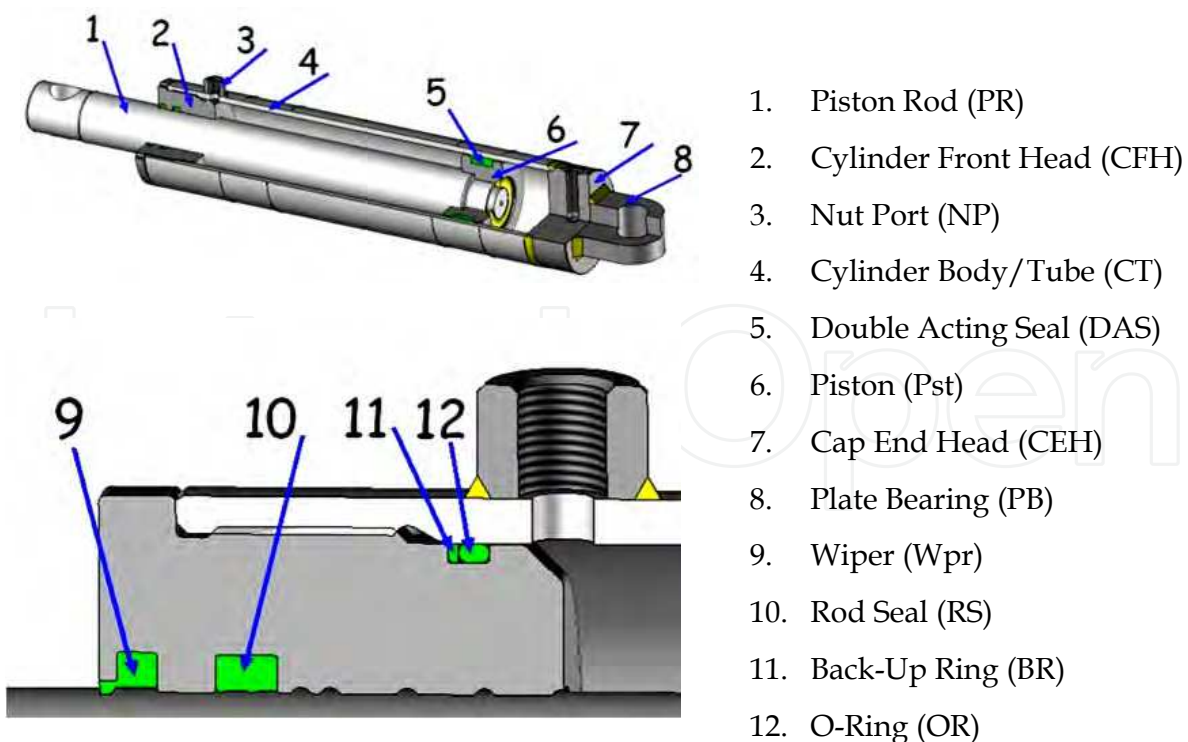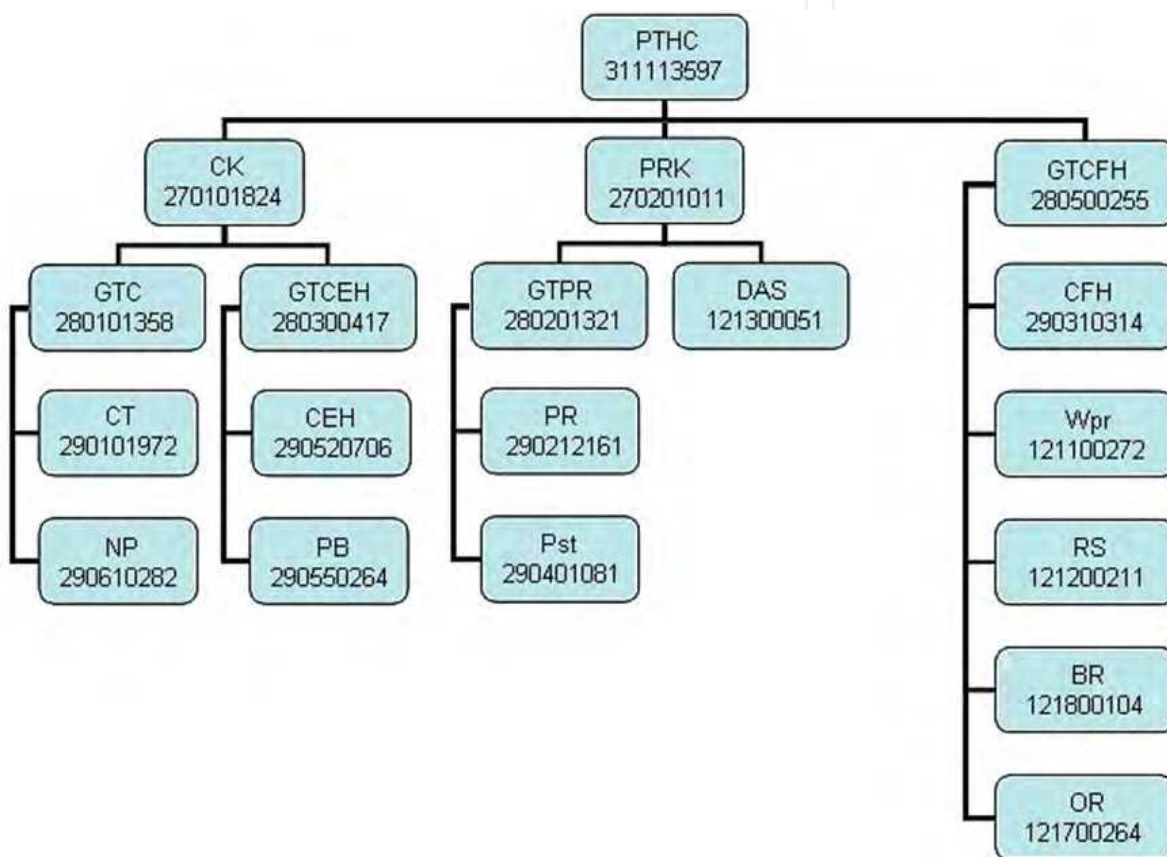11. Back-Up Ring (BR)
12. O-Ring (OR)

Fig. 1. Components of a hydraulic cylinder

The parts of the hydraulic cylinder are constructed in a precise manner. Fig. 2 shows the structural block diagram of hydraulic cylinder's manufacturing. Each block of the structure incorporates the abbreviations of the parts and the digital identification numbers, by which they are classified in the product management system. These numbers are unique for each object. The given block diagram shows that the PTHC (Hydraulic Cylinder of Piston Type) is located at the highest organizational level. It represents the final product which incorporates all components of the hydraulic cylinder structure. In the production management system PTHC exists under the digital identification number 311113597, which specifies its particular classification. Through it, the cylinder is controlled by the design and operation systems.

DAS – Double Acting Seal
CT – Cylinder Tube
PR – Piston Rod
Pst – Piston
CFH – Cylinder Front Head
CEH – Cap End Head
PB – Plate Bearing
NP – Nut Port
OR – O-Ring
BR – Back-Up Ring

RS – Rod Seal
Wpr – Wiper
GTC – General Type Cylinder
GTPR – General Type Piston Rod
GTCFH – General Type Cylinder Front Head
GTCEH – General Type Cap End Head
CK – Cylinder Kit
PRK – Piston Rod Kit
PTHC –Hydraulic Cylinder of a Piston Type

Fig. 2. Structural block diagram constituting a hydraulic cylinder

• The studied PTHC is assembled through the following few steps:
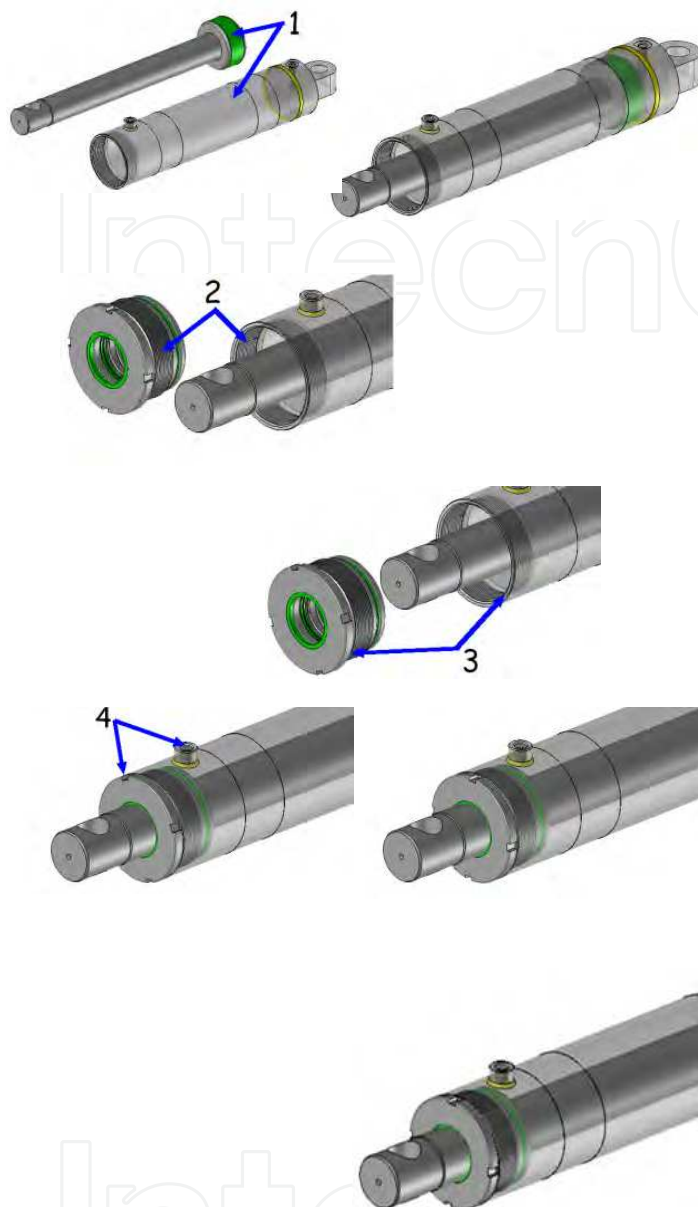
Step 1 –setting the mates between the axes of the Piston Rod Kit and of the cylinder. Thus, the relative degrees of freedom of the PRK to cylinder body are set to 2: longitudinal translation and rotation around its axis.

Step 2 – setting mates between the axis of the cylinder front head and the axis of the cylinder. The axis of the cylinder front head is automatically alined towards the cylinder axis.

Step 3 – setting „Face to Face"the front surface of the cylinder front head and the front surface of the cylinder. The cylinder front head automatically touches the front of the cylinder.

Step 4 – orientating the cylinder front head towards the nut port opening and establishing the final PTHC appearance.

Fig. 3. Assembling of a hydraulic cylinder of a piston type (PTHC)

4.4 SolidWorks (SW) Flow Simulation environment

SW Flow Simulation package is designed for simulation of fluid and gas phenomena. It enables development of various scenarios for modeling different fluids and simulating their floating, including calculating of speed, acceleration, turbulence, etc. of fluid particles, of the forces on the surrounding walls, of the heat exchange effects, etc. The library set of examined fluids varies starting from air and water through gas and liquid chemicals to ice cream paste, honey, plastic melts, blood, etc. The user can also create user-defined fluids. Some of the main features of SW Flow Simulation enable modeling of: heat exchange effects with conduction or convection; heat exchange through radiation, including calculations on solar radiation; fluid simulation of gas and fluid flows in valves, regulators and pipes;

simulation of complex rotating flows in mechanisms. Additional SW Flow Simulation options enable: investition on externally wrapped moving objects; analysis of turbulent flows and illustration of trajectories of turbulating particles; simulation of real gasses for more precise analysis of high pressure or low temperature problems for example identification of domains where cavitation is possible and so on.

4.5 SW Flow Simulation analysis of the modeled hydraulic cylinder

Through this simulation the design of the real working environment of the hydraulic cylinder under a specified variety of boundary conditions, real time monitoring the motion of the fluid inside the cylinder, examination of the magnitude of the force between the plunger and the cylinder walls for different working volumes of the two cylinder chambers and systematizing the numerical and graphical results of all performed numerical experiments are done.

Depending on a predefined scenario of hydraulic cylinder work SW Flow Simulation software tracks the fluid flow trajectories inside the volumes of the hydraulic cylinder, regarding their development with time. All mathematical procedures are influenced by the boundary conditions and the scenario goals and are described below.

Through the first stage a mathematical definition of the investigated volume is done. The local coordinate systems inside each examined chamber volume are introduced and the computing domain is outlined. The disposition of the fluid, particularly its location, form and type (viscosity, density, operating temperature, etc.) is also input.

The boundary conditions and the type of mathematical calculation, including choosing the solver, are adjusted at the second stage. Thus, working environment is described.

The numerical and visual presentations (graphical or as a movie) are the focus of the third stage. All obtained data is systematized by the software and can be analized by the user.

To enable SW Flow Simulation to run the simulation and to set goals correctly, it is necessary to specify the material of each part of the model. Since the provided hydraulic cylinder is a product system of "HES" PLC, Jambol, all material characteristics correspond to the technical data given in its technical documentation. The structured parts are made of carbon steel. The materials of all non-structural parts are also defined.

To perform the research a new project is started. The SI measuring system is chosen.

After that the following characteristics of the "internal" fluid are specified: location in a bounded space; no cavities; positive gravity direction - along +Y axis; axis of motion - Y. Other applicable options like heat conduction, radiation, etc are not considered in this case.

Next, the type of the fluid is chosen. It is oil of type ISO-L-HL, according to standart system ISO 6743/4, also known as MH-L32. It posesses the following characteristics: viscosity class - 32; density at 20˚C - 0,874 g/ml; kinetic viscosity at 40˚C - 32 mm²/s; viscosity index - 96; liquefaction temperature - -21˚C. As there is no such type of fluid in SW database, a new one is assigned.

The next few steps implement computational domain function (fig 4-left), fluid subdomains (fig. 4-right), boundary conditions (fig. 5) and computational goals. The precise introduction of computational domain is of significant importance for its

insufficient volume may cause inaccurate results, while its unreasonable increase "aggravates" the model and extends the necessary CPU time. The surrounding surfaces of inside chambers are highlighten to outline the fluid subdomains. The two volumes should be totally separate, should have no common overlap and should be completely bounded. Among the defined goals are the factors, which influence the system operation, such as the pressure magnitude, including its dynamic and static components. „Flow Trajectories"command is used to track the motion of the fluid particles. Obtained data can be exported to files of the following types avi, excell or pictures (*.jpg or others).

Computational domain   Fluid sub-domain
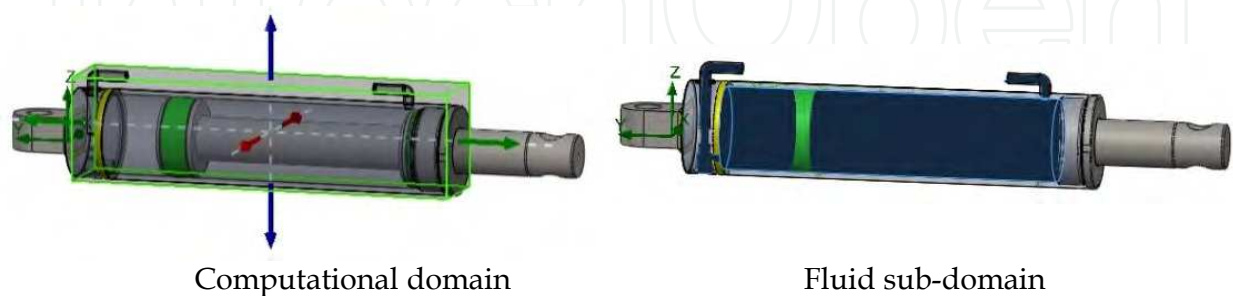
Fig. 4. Introducing the of the examined fluid domains

„Pipe" domain   „Chamber" domain
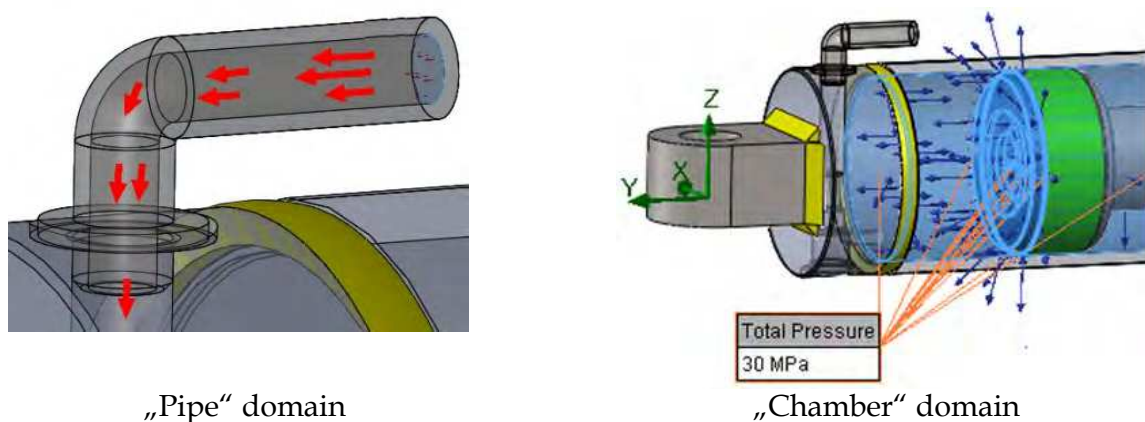
Fig. 5. Defining the boundary conditions of the incoming fluid

During the first stage of the simulation the piston rod kit (PRK) moves forward.

To define the boundary conditions of the incoming fluid the following steps are passed through:

- For the the „pipe" domain (fig. 5 - left):
  1. Specifying the opening (area, surface) through which the fluid fills the pipe.
  2. The direction of fluid motion is selected.
  3. "Flow Openings" function is introduced by specifying properties (weight, volume or speed) of the fluid filling the pipe. In our case this is the specific flow rate. The required flow rate for the designed hydraulic cylinder is 35 l/min or 0,00058 m^3/s.
- For the the „chamber" domain (fig. 5 - right):
  4. Highlightening all the inside walls which outline the first volume and which are exposed to pressure.

5. Highlightening the direction of the rod motion - Y.
6. Selecting "Pressure Openings" function and specifying surfaces, exposed to pressure. For this project "Total Pressure" option is chosen and the pressure value is 30MPa. This is the test pressure, according to hydrauluc cylinder's technical documentation.
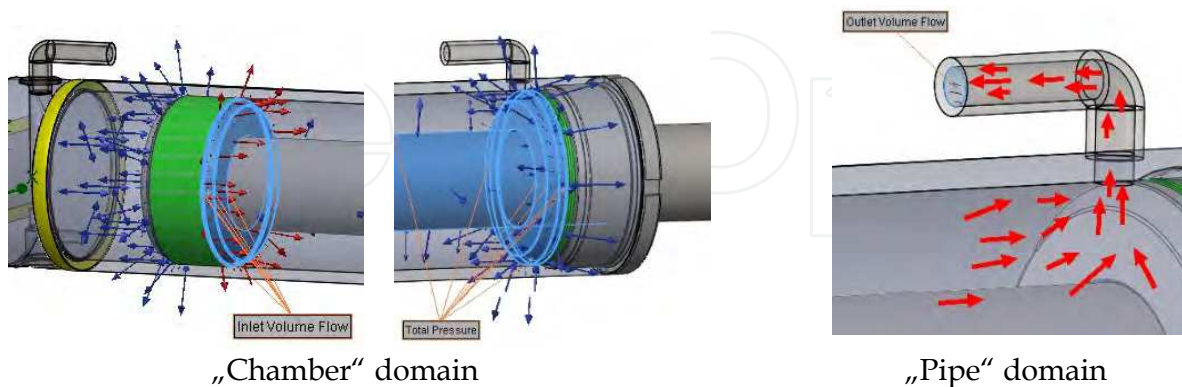7. The temperature of the fluid is set at 50°C.

„Chamber" domain   „Pipe" domain

Fig. 6. Defining of boundary conditions for the outgoing fluid

Boundary conditions of the outgoing fluid (fig. 6) are set, depending on following requirements:

1. To visualize the simulation of the fluid going out of the right chamber, it is obligatory to define the boundary conditions. While moving, the piston pushes the fluid out of the right chamber. The pressure resulting from the fluid expulsion is introduced throught highlightening the affected surfaces. All these actions are similar to the ones described above.
2. The boundary conditions at the fluid exit are input about the inner surface of the pipe. The procedure "Outlet Volume Flow" is used.

Thus all boundary conditions are precisely defined.

The next step is to define the "Goals" of the output, i.e. the results that are going to be analysied by the user later. The aim of the study is to find the value of the force, raised by fluid and moving the piston.

5. Numerical data and results obtained through SW Flow Simulation for the modeling hydraulic cylinder

5.1 Moving forward of the piston rod kit

5.1.1 Basic assumptions

The course of the piston of the hydraulic cylinder is 310 mm (in both directions). This means that the distance L varies from 0 ÷ 310mm (fig. 7). As the used software is unable to solve the system for the three processes (filling the left chamber, moving the piston and empting the right chamber) going on simultatiously the following assumption is made: The system is solved for some close chain steady-state situations while the initial boundary conditions for each situation coincide with the final boundary conditions of the previous situation.

To enable calculation of the magnitude of force $\vec{F}$ (fig.7) depending on the piston's position, the operating volume of the left chamber of the hydraulic cylinder increases as distance L increases and the simulation is run for each particular position of the piston.

Fig. 7. Operating volume of the hydraulic cylinder

As the formulation and the goals of the problem are defined the calculation should be run.

5.1.2 Visualizing the results and comments on the simulation process

A preview of operating hydraulic cylinder during forward piston course is shown in fig. 8. The PRK tries to move forward in *Vrod* direction, while squeezing forward the outgoing fluid. Consequently the volume of the right chamber shrinks.

Visualized as separate vectors flow trajectories enable easier tracking of the motion of fluid particles (fig. 8 and 9).

Boundary conditions   Flow trajectories
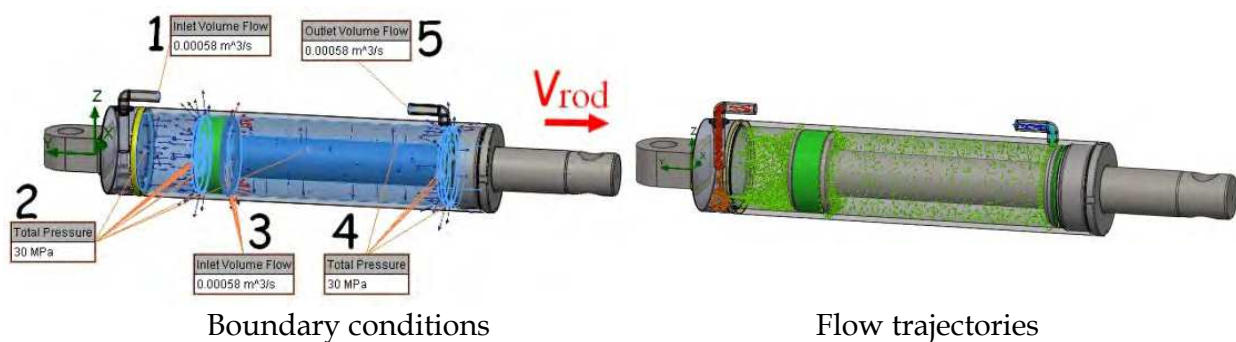
Fig. 8. A preview of an operating hydraulic cylinder – PRK moving forward

Fluid filling the chamber   Fluid empting the right chamber   Legend
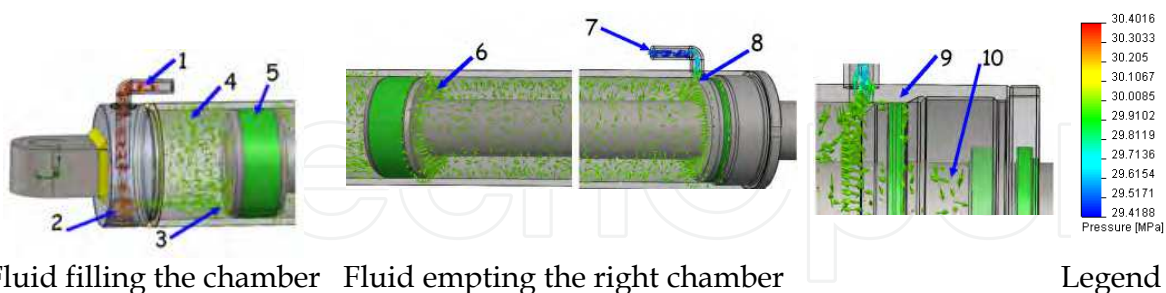
Fig. 9. Flow trajectories of fluid particles inside the hydraulic cylinder for L= 65mm.

- Position 1 – The working fluid fills the volume through a pipe of a diameter of 8 mm with a flow rate Q = 35 l/min.
- Position 2 – The fluid particles form a small vortex at the bottom and track forward through an opening of 6mm diameter. For the presented situation the volume of the left cylinder chamber is $V = 0,000413 m^3$ and is filled for t=0,71s.
- Position 3 – The velocity of the fluid filling the left cylinder chamber as a strongly directed stream, is about 20,6 m/s.

- Position 4 – Some of the fluid particles re-bounce off the piston and vortex around. According to software calculation, their speed is approximaely 10m/s.
- Position 5 – PRK separating the two volumes.
- Position 6 – Moving forward the piston initiates the fluid expulsion from the right chamber. The pressure of the fluid is about 29,95 MPa. The fluid particles try to squeeze the piston head and press the DAS.
- Position 7 –The fluid particles slide throught the chamfer of the cylinder front head and float up the pipe with a velocity of about 11,6 m/s and pressure of about 29,7MPa.
- Position 8 – The fluid leaves the computational domain through the pipe, which diameter is 8 mm. The pressure inside this domain decreases slightly to about 29,4 MPa.
- Position 9 – Some of the fluid particles have passed through the joint between the cylinder front head and the cylinder and have disposed around the ring.
- Position 10 – Some other particles of the fluid have disposed in the volume surrounded by the rod surface and the screw channel of the cylinder front head, re-bouncing off the rod seal.
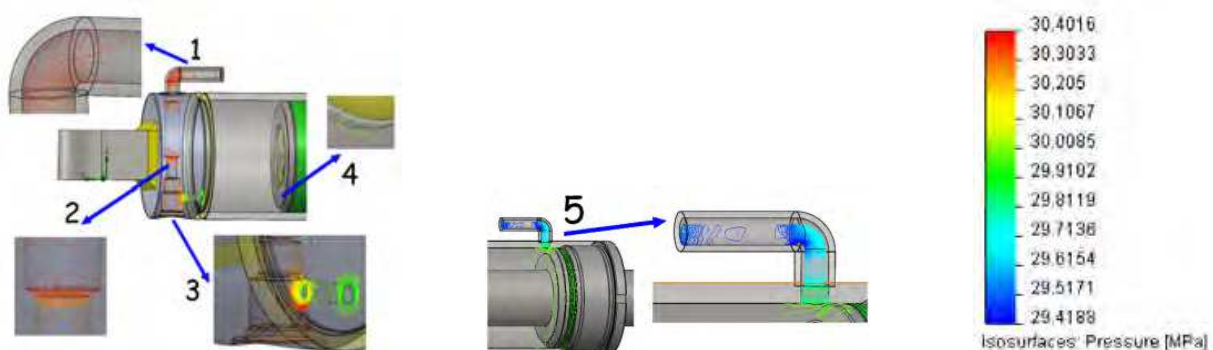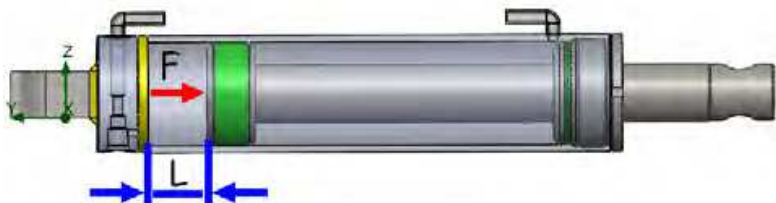
Fig. 10. Dangerous domains in the hydraulic cylinder chambers

To define the dangerous domains during hydraulic cylinder operation, i.e. the domains where the flows pressure is high, "Isolines" function is used - fig. 10. This figure enables user to find the potential crack originators in the cylinder structure.

- Position 1 – The fluid fills the pipe with high velocity. It is possible for cracks to be originated at the corner of the pipe.
- Position 2 – The fluid floats through a zone chamfering pipes with different diametres.
- Position 3 – The high pressure squeezing the fluid inside the hydraulic left cylinder chamber can cause vulnerability around this zone.
- Position 4 – The fluid particles re-bounce off the piston of the highest velocity. The chamfered piston profile is exposed to random time-varying pressure.
- Position 5 – The fluid is released at a high speed, cracks are possible to occur in the joint between the cylinder tube and the pipe.
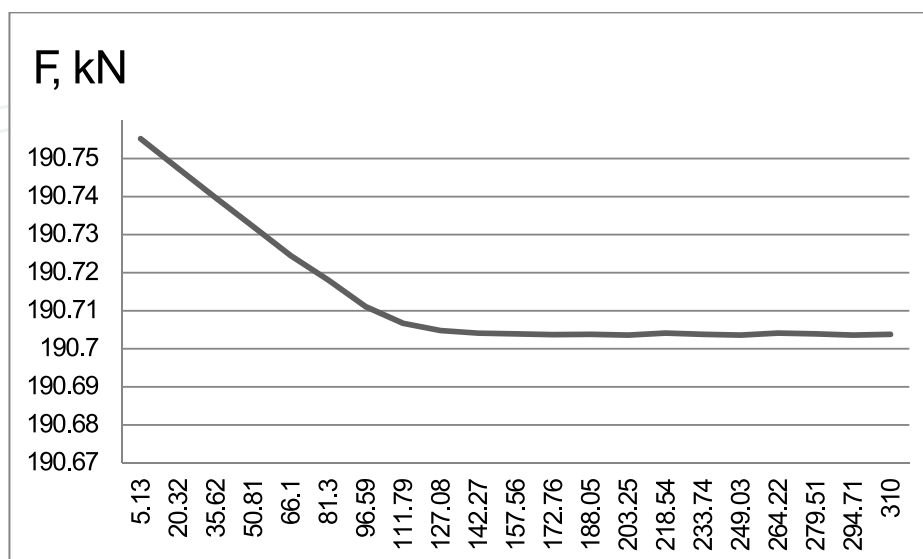
5.1.3 Numerical data and verification of the results

All numerical results of any importance to the study are exported to MS Excel file. Some of them are systematized in table 1 and graphically presented in fig. 11. During the second half of the piston's forward course the magnitude at first decreases slightly and then remains constant.

№	L, [mm]	V, [m³]	t, [s]	Vr, [mm]	F, [kN]	Legend
1	5	3,179E-05	0,055	5,13	190,7552	
2	20	1,272E-04	0,218	20,32	190,7474	**L** - the distance defining PRK
3	35	2,225E-04	0,382	35,62	190,7397	position inside the cylinder;
4	50	3,179E-04	0,545	50,81	190,7321	
5	65	4,133E-04	0,709	66,10	190,7245	**V** - the volume of the left
6	80	5,087E-04	0,872	81,30	190,7181	cylinder chamber;
7	95	6,041E-04	1,036	96,59	190,7111	
8	110	6,994E-04	1,199	111,79	190,7067	**t** - the time for filling the
9	125	7,948E-04	1,363	127,08	190,7048	chamber of the hydraulic
10	140	8,902E-04	1,526	142,27	190,7041	cylinder with fluid;
11	155	9,856E-04	1,690	157,56	190,7039	
12	170	1,081E-03	1,853	172,76	190,7037	**Vr** - the displacement of PRK
13	185	1,176E-03	2,017	188,05	190,7038	in forward direction during
14	200	1,272E-03	2,180	203,25	190,7036	its motion;
15	215	1,367E-03	2,344	218,54	190,7041	
16	230	1,462E-03	2,507	233,74	190,7038	**F** - the calculated by the SW
17	245	1,558E-03	2,671	249,03	190,7036	Flow Simulation magnitude
18	260	1,653E-03	2,834	264,22	190,7041	of the tracked force during
19	275	1,749E-03	2,998	279,51	190,7039	the forward motion of PRK;
20	290	1,844E-03	3,161	294,71	190,7036	
21	305	1,939E-03	3,325	310,00	190,7038	

Table 1.

Fig. 11. Graph of the magnitude of force $\vec{F}$ versus forward displacement of the piston **Vr**

The calculated magnitude of the force $\vec{F}$ is verified through the basic methods of Fluid Mechanics, as follows:

- test pressure of the hydraulic cylinder - $P = 30MPa$;

- the area at "face" surface of the piston - $A = \dfrac{\pi . D^2}{4} = \dfrac{3,14.90^2}{4} = 6358,5mm^2$

- calculated force - $F = 30.6358,5 = 190755N \Rightarrow 190,755kN$

- error - $\Delta\% = \dfrac{190,755 - 190,713}{190,755}100\% = 0,022\%$

The error is under 0,025%, where the value 190,755 is the theoretically calculated force magnitude and the value 190,713 is the average value of the results run by the software.

5.2 Moving backwards of the piston

5.2.1 Basic assumptions and boundary conditions

The method and the stages of simulating the backward motion of PRK are the same as they are during its forward movement simulation but the set boundary conditions are different.

Fig. 12 shows the sequence of setting the input boundary conditions.

Step 1 – The opening where the operating fluid enters the computational domain is specified. This is the pipe through which the fluid fills the right chamber of hydraulic cylinder.

Step 2 – All walls surrounding the right cylinder chamber are exposed to pressure of 30 MPa.

Steps 3 and 4 – Defines the boundary conditions of the fluid expulsed out of the left cylinder chamber.

Step 5 – Setting the boundary conditions at the fluid outlet.

Fig. 12. Setting the boundary conditions during PRK backward motion
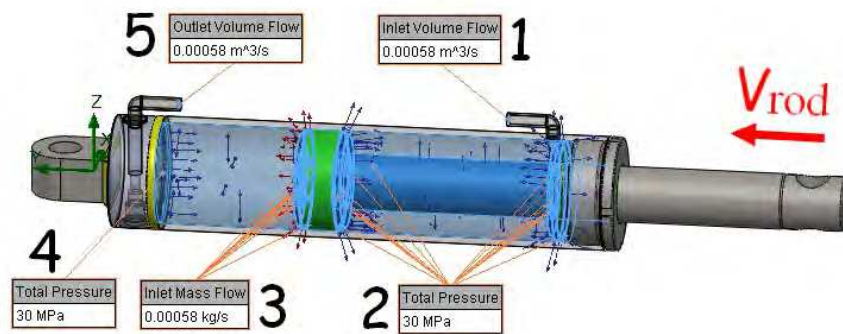
Fig. 13. Scheme of the operating cylinder during piston's backward course

Scheme of the operating cylinder during piston's backward course is given in fig. 13.

- Position 1 – The fluid fills the right chamber of the hydraulic cylinder throught the pipe opening at a flow rate of 0.00058m³/s
- Position 2 – The inside pressure of 30 MPa is established.
- Position 3 – PRK moves backwards in Vrod direction, the volume of the left chamber shrinks while the piston expulses the fluid.
- Position 4 – The inside pressure of the left cylinder chamber is 30 MPa.
- Position 5 – The fluid inside the left cylinder chamber is expulsed.

5.2.2 Flow trajectory and isoline results of the run simulation

Fig. 14-right shows the flow trajectories in the right chamber of the hydraulic cylinder at L = 185 mm (see table 2).

- Position 1 – The operating fluid fills the domain through an 8mm pipe at a flow rate of Q= 35l/min ($0,0005833m^3/s$).
- Position 2 – The fluid fills the cylinder chamber as a highly concentrated stream. Some of its particles rebounce off the chamber of the cylinder front head and the piston rod. The velocity of the fluid at that domain is about 11,6 m/s.
- Position 3 – The fluid fills the chamber, while some of its particles rebouce off the surface of the piston. The right cylinder chamber is filled for about t = 1,39s.
- Position 4 – The fluid flows around the piston rod and fills the entire right chamber.

The motion of the fluid particles inside the left chamber of the hydraulic cylinder is shown in fig. 14-left.

- Position 1 – The back side of the piston pushes the fluid forward with a pressure of about 29,94MPa. The fluid is attempting to drain through the DAS to the right cylinder chamber.
- Position 2 – Some of the fluid rebounce off the caped head.
- Position 3 –The fluid empties the chamber of the hydraulic cylinder through a 6mm opening. The pressure in this domain is about 29,5MPa and the velocity is around 20,6m/s.
- Position 4 –The fluid goes out through the hole at the bottom.
- Position 5 – The fluid leaves the computational domain.

The general view of the flow trajectories inside the hydraulic cylinder is given in fig. 15.

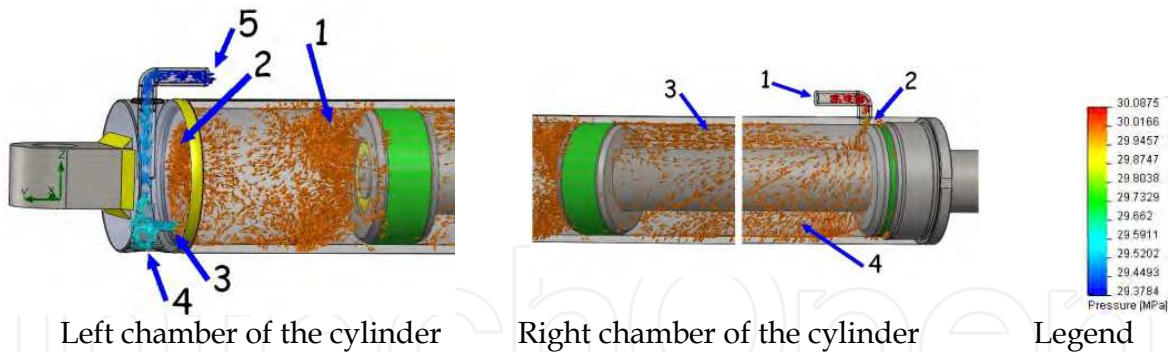Left chamber of the cylinder   Right chamber of the cylinder   Legend

Fig. 14. Flow trajectories inside the hydraulic cylinder at L = 185 mm (see table 2).

Fig. 15. General view of the flow trajectories inside the hydraulic cylinder

"Isolines" function helps the designers to study in details the vulnerable zones of the hydraulic cylinder structure (fig. 16).

- Position 1 – The fluid fills the right chamber of the hydraulic cylinder at the highest velocity and occurrence of cracks in the pipe corner is possible.
- Position 2 – The fluid empties the cylinder through the left chamber pipe.
- Position 3 – The joint between the cylinder chamber and the pipe is among the vulnerable cylinder zones too. Because of the high pressure inside this domain and the chamfered joints crack occurence is possible.
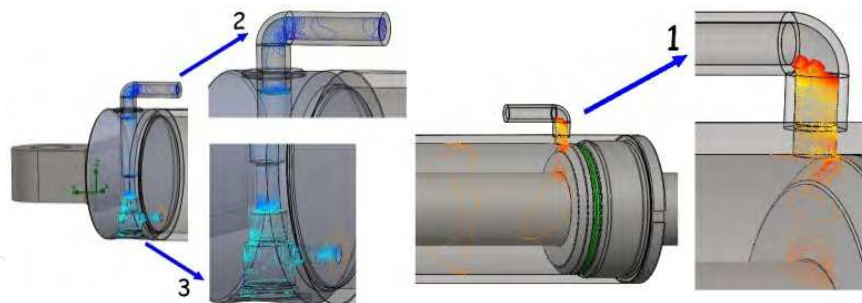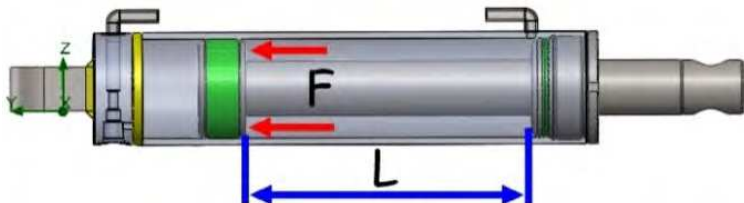
Fig. 16. Vulnerable domains of the hydraulic cylinder structure during PRK backward motion

5.2.3 Numerical data and verification of the results

The numerical and graphical presentations of the magnitude of the studied force $\vec{F}$ during piston's backward motion are given below (table 2 and fig. 17)

The calculated magnitude of the force $\vec{F}$ is verified through the methods of Fluid Mechanicsm, once more:

- test pressure of the hydraulic cylinder - $P = 30 MPa$;
- the area of the piston's "face" surface - $A_{piston} = \dfrac{\pi.D^2}{4} = \dfrac{3,14.90^2}{4} = 6358,5 mm^2$

№	L, [mm]	V, [m³]	t, [s]	Vr, [mm]	F, [kN]	Legend
1	5	2,198E-05	0,038	5,13	131,8805	
2	20	8,792E-05	0,151	20,37	131,8668	L - the distance defining PRK
3	35	1,539E-04	0,264	35,61	131,8576	position inside the cylinder;
4	50	2,198E-04	0,377	50,86	131,8505	
5	65	2,857E-04	0,490	66,10	131,8466	V - the volume of the right
6	80	3,517E-04	0,603	81,34	131,8453	cylinder chamber;
7	95	4,176E-04	0,716	96,59	131,8448	
8	110	4,836E-04	0,829	111,83	131,8445	t - the time for filling the
9	125	5,495E-04	0,942	127,08	131,8447	chamber of the hydraulic
10	140	6,154E-04	1,055	142,32	131,8448	cylinder with fluid;
11	155	6,814E-04	1,168	157,56	131,8446	
12	170	7,473E-04	1,281	172,81	131,8447	Vr - the displacement of PRK
13	185	8,133E-04	1,394	188,05	131,8446	in backward direction
14	200	8,792E-04	1,507	203,29	131,8448	during its motion;
15	215	9,451E-04	1,620	218,54	131,8445	
16	230	1,011E-03	1,733	233,78	131,8447	F - the calculated by the SW
17	245	1,077E-03	1,846	249,03	131,8445	Flow Simulation magnitude
18	260	1,143E-03	1,959	264,27	131,8446	of the tracked force during
19	275	1,209E-03	2,072	279,51	131,8448	the backward motion of
20	290	1,275E-03	2,185	294,76	131,8447	PRK;
21	305	1,341E-03	2,298	310,00	131,8445	

Table 2.
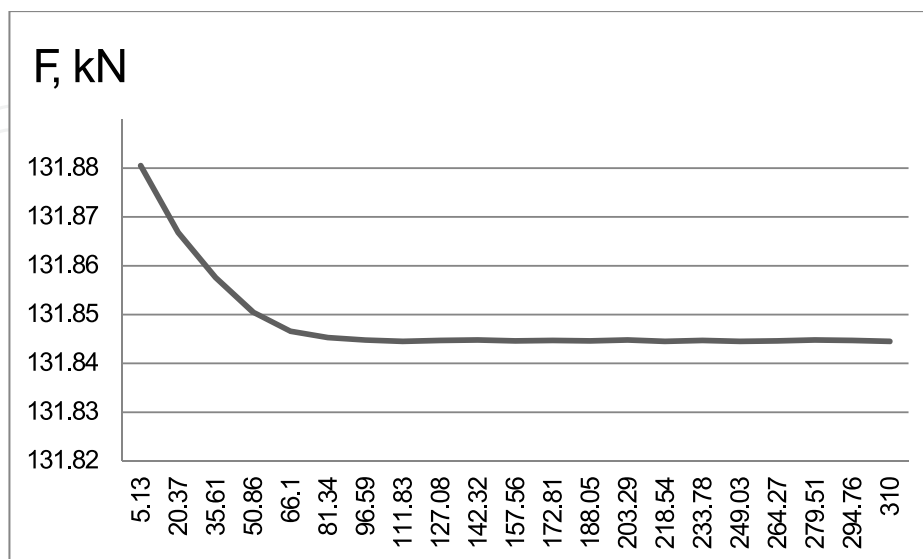
Fig. 17. Graph of the magnitude of force $\vec{F}$ versus backward displacement of the piston **Vr**

- the area of the rod - $A_{rod} = \dfrac{\pi.D^2}{4} = \dfrac{3,14.50^2}{4} = 1962,5mm^2$.

- the "working" area of the piston - $A = A_{piston} - A_{rod} = 6358,5 - 1962,5 = 4396mm^2$

- calculated force - $F = 30.4396 = 131880N \Rightarrow 131,880kN$

- error - $\Delta\% = \dfrac{131,88 - 131,85}{131,88}100\% = 0,023\%$

The error is under 0,025%. It is calculated as the theoretical force magnitude is equal to 131,88 kN and 131,85 is the average value of the results run by the software.

Based on the given in fig. 17 graph it can be concluded that the magnitude of the force $\vec{F}$ decreases slightly in the second half of the course motion. The tendency in madnitude decrease corresponds to the one observed during the pistons course forward. The difference in magnitude values is due to gravity impact and to the difference in the "working" area of the piston.

6. CAE simulation of the operational movement of a hydraulic cylinder using the SW Motion environment

6.1 Developing a SW motion scenario

The SW Motion software enables performing a real time dynamic investigation on the phonomena in the hydraulic cylinder. The input data is predefined in our CAD model and SW Flow Simulation research.

The objectives of this part of the research are:

- to simulate the dynamic effects of the piston motion under predefined input data and various boundary conditions;
- to calculate through a numerical experiment and to analyse the friction force arising between the inner surface of hydraulic cylinder and the double acting seal (DAS).

At first a design scenario, including all facts, which influence the cylinder operation and the calculating steps is defined. The direction and the motion velocity are introduced. All necessary data is taken from performed SW Flow Simulation research. The intermediate values are calculated automatically through cubic interpolation.

The second step is to define the contact conditions between the inner surface of the cylinder and the outer surface of the DAS. Here the characteristics of the materials are input. A greasy environment is chosen. Based on user's choice, the program itself calculates coefficients of static and kinetic friction between the materials.

During the third step a time dependent calculating intervals are input. After starting the "Solve" procedure, all kinematic and dynamic parameters of the motion are automatically calculated. During this last step all results selected by the user are systematized to enable easier analysis.

It is necessary to add some additional explanations about the second and the third steps of the model development.

The materials which form the seal (fig. 18) are different, but they all belong to polyester elastomers'group. Elastomers are polymeric materials that can be subjected to reversible high-elasticity deformations in extreme operating conditions. They possess an extreme abrasion and extrusion resistance. Most often the piston seals are made of Turcon materials or Zurcon polyurethane. In our case, the material of the whole seal is defined as "Greasy", i.e. working in viscous environment. Thus, the coefficients of static and kinetic friction (μ_s

and μ_k) are automatically found by the software.

          position 1   position 2   legend
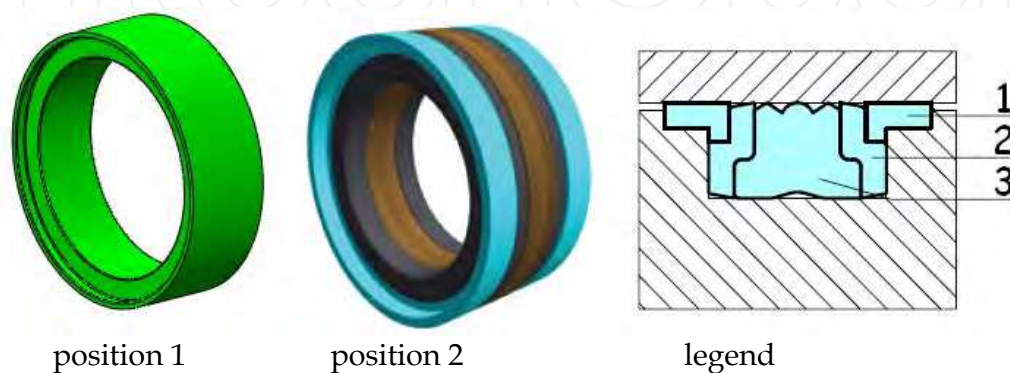
Fig. 18. Double acting seal

Real time interval of the simulation for forward motion of PRK is equal to 3,325s (see the previous item). The backward motion scenario developes analogically, but the process lasts 2,298s.

6.2 Analysis of the results obtained through the developed SW Motion Scenario

The motion of the piston is stored in visual file of the type *. avi, which is easily transferred from one working environment to another. Some snapshots of the forward and backward motion of PRK are given in fig. 19.

In the final simulation stage the graphs of some studied parameter versus time are displayed. The software offers a lot of types of results, such as "Force", "Displacement", "Velocity", "Acceleration", "Momentum", "Energy", "Power", etc. The most important for the investigation function is classified in "Force" group. It includes "Applied Force", "Reaction Force", "Reaction Momentum", etc. The authors are interested in "Contact Force", which tracks the magnitude of the friction force between the cylinder and the DAS (fig. 20) during the simulation. The friction force graph versus time is given in fig. 21.

As the authors could not find any research on the topic of how the contact force between the cylinder and the DAS can be calculated, they have decided to verify the proposed methodology of calculating the friction force as follows: there are some measurements made by the producer "HES" PLC, Jambol of the pulling/pushing force on the piston in hydraulic cylinder of that size and construction. The contact force, arisen during PRK motion due to its dead weight, varied in-between 20-30N. The simulated force varied in the range of 22-28N (see fig. 21). The time duration of rod piston course has been chosen to be the same as if the hydraulic cylinder is filling with a fluid of a pressure of 30MPa.

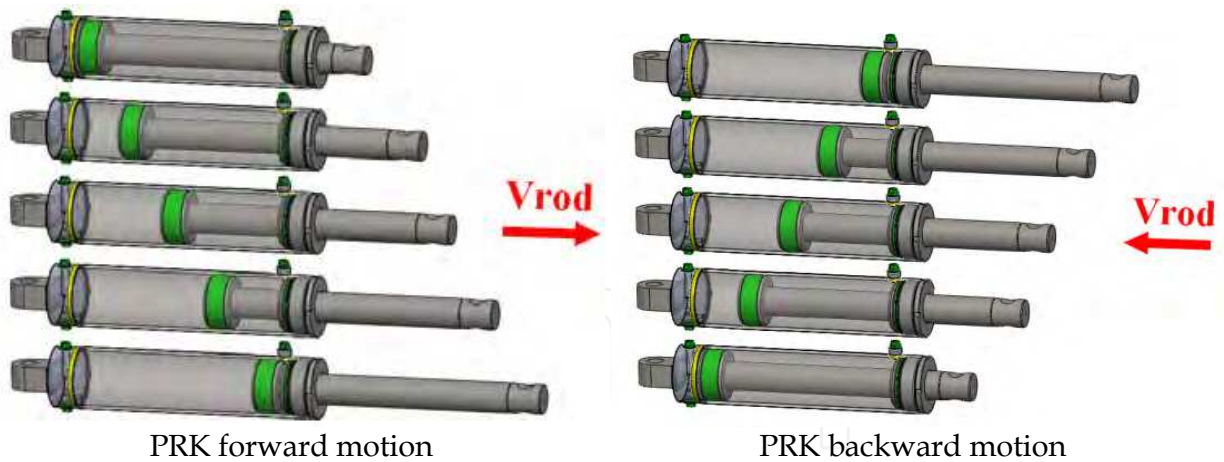PRK forward motion   PRK backward motion

Fig. 19. Some snapshots of the motion of the piston rod kit
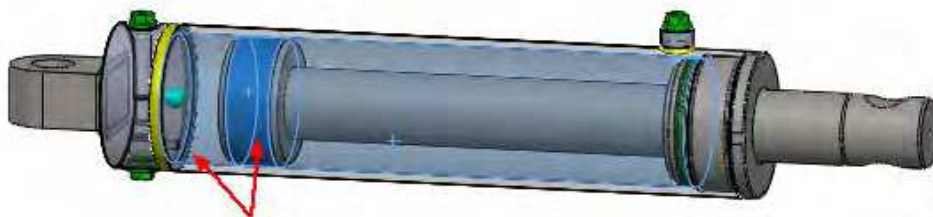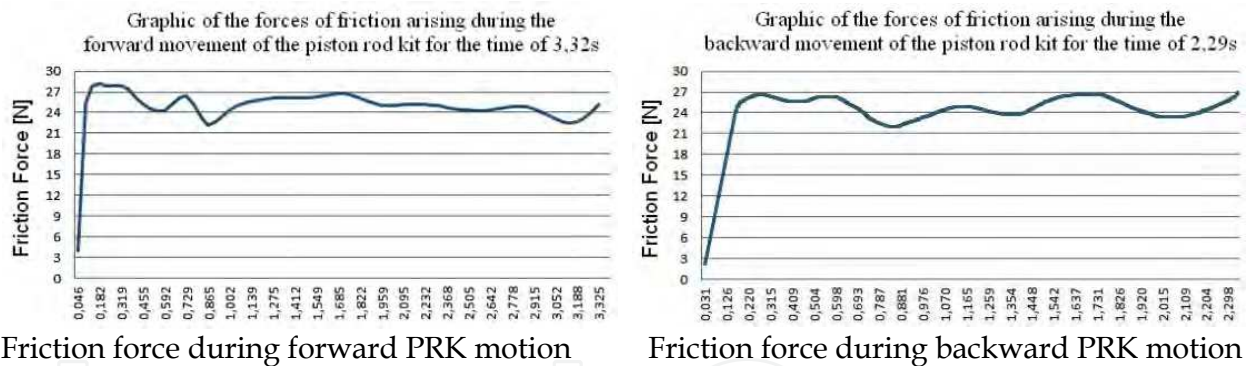

Fig. 20. The surfaces between which the friction force is tracked

Friction force during forward PRK motion   Friction force during backward PRK motion

Fig. 21. Graph of the friction force during the motion of the piston rod kit

7. Conclusions

The developed integrated technology of CAD modeling and CAE analysis of a basic hydraulic cylinder is a part of the long-lasting strategy of company "HES" PLC, Jambol for increase of quality and production control, for bettering of the working environment, etc. Among the basic objectives of this strategy are:

- Improvement of the productivity of three-dimensional modeling of hydraulic cylinders as a result of the established structural scheme and easier and quicker design of hydraulic cylinders' particular elements.
- Development of reliability and overall structural and technological optimizing procedures of the designed hydraulic cylinders, supported by integrated technology for CAE analysis of the fluid flows through FVM.

- Increase of quality and control of the produced products, throughout the use of competitive technology targeting stronger integration of CAD/CAE/CAM systems for product design.

All these stages result in a new working environment, which provokes designers' creativity, optimises technology, increases the competitiveness of the company and finally, strengthens its positions on the market of hydraulic cylinders.

8. Acknowledgment

9. References

Brenan, K.E., Campbell, S.I., and Perzold, L.R. (1996). Numerical Solution of Initial Value Problems in Differential-Algebraic Equations, Classics in Applied Mathematics, ISBN 0-89871-353-6

Gear, C.W. (1971a). The Simultaneous Solution of Differential Algebraic Systems. IEEE Transactions on Circuit Theory, CT-18, No.1, 89-95.

Gear, C.W. (1971b). Numerical Initial Value Problems in Ordinary Differential Equations. New Jersey: Prentice-Hall.

Lombard Matt (2011). SolidWorks 2011 Assemblies Bible, John Wiley and Sons, ISBN10: 1118002768

Lombard Matt (2011). SolidWorks 2011 Parts, John Wiley and Sons, ISBN10: 111800275x

SolidWorks Flow Simulation 2011 Technical Reference, (2011), Dassault Systèmes SolidWorks Corp.

Solving Engineering Problems with Flow Simulation 2011, (2011), Dassault Systèmes SolidWorks Corp.

SolidWorks Motion Studies, on-line reference guide, (2011), Dassault Systèmes SolidWorks Corp.

Versteeg Henk Kaarle, Weeratunge Malalasekera (2007). An introduction to computational fluid dynamics: the Finite Volume Method, Pearson Education Limited, ISBN 978-0-13-127498-3

Wendt John F., John David Anderson (2009). Computational fluid dynamics: an introduction, Springer-Verlag, Berlin, ISBN 978-3-540-85055-7

Топалова М., Ст. Бакърджиев (2006). 3D моделиране на технологични единици, Известия на СУБ – Сливен, т. 10, 2006, с. 49-51, ISSN 1311-2864.

Янакиев И., Ст. Николов (2010). CAD/CAM/CAE системи в машиностроенето, ISBN 978-954-737-802-5
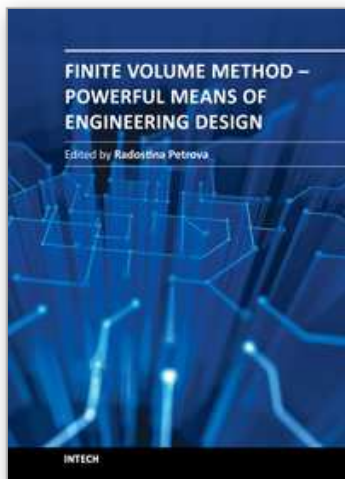
Finite Volume Method - Powerful Means of Engineering Design

Edited by PhD. Radostina Petrova

ISBN 978-953-51-0445-2

Hard cover, 370 pages

Publisher InTech

Published online 28, March, 2012

Published in print edition March, 2012

We hope that among these chapters you will find a topic which will raise your interest and engage you to further investigate a problem and build on the presented work. This book could serve either as a textbook or as a practical guide. It includes a wide variety of concepts in FVM, result of the efforts of scientists from all over the world. However, just to help you, all book chapters are systemized in three general groups: New techniques and algorithms in FVM; Solution of particular problems through FVM and Application of FVM in medicine and engineering. This book is for everyone who wants to grow, to improve and to investigate.

INTECH
open science | open minds