Journal of Physics: Conference Series **1426** (2020) 012005 doi:10.1088/1742-6596/1426/1/012005

Piston crank mechanism simulation using finite element method

I Zs Miklos, C C Miklos and C I Alic

Politehnica University of Timisoara, Department of Engineering and Management, 5 Revolution Street, Hunedoara, 331128, Romania

E-mail: imre.miklos@upt.ro

Abstract. In this paper we present the analysis of the piston crank mechanism, using the finite element method using mechanical event simulation (MES). Mechanical event simulation is a modern process of modeling and analysis of mechanical systems, much different from the classical methods known in mechanical engineering. The use of MES involves a combination of kinematics of mechanical systems, dynamics of rigid bodies, respectively nonlinear stress analysis. So, mechanical event simulation is engineering by simulating a physical event in a virtual laboratory. The crank-piston mechanism analysis has been achieved with the Simulation Mechanical program, which aimed to determine the stress state, nodal displacement or relative elongations (strains) of mechanism elements, at different time steps of the kinematic cycle, depending on the kinematic and kinetostatic parameters variable according to different curves.

1. Introduction

The piston-crank mechanism is a mobile mechanical system, mainly found in the internal combustion engines of road vehicles. It consists of kinematic elements as crank, connecting rod and piston (slider) and kinematic couplings of various classes with the role of limiting the degrees of freedom of these elements. If the crank is the leading element, the mechanism is designed to convert its rotary motion to straight-line motion of the piston, or vice versa, if the piston is the leading element with internal combustion engines.

The piston crank mechanism is a planar mechanism, the study of which involves performing an analysis of it, by means of which we determine the size of kinematic components, position, velocity and linear or angular acceleration depending on the type of their movement, kinetostatic sizes respectively, such as forces and moments of forces which are acting on the elements and kinematic couplings, for as many successive positions as possible of the leading element.

In theory and practice, there are several procedures to analyze mechanisms, such as graphical and grapho-analytical methods, and analytical methods as well. Graphic and grapho-analytical methods involve high amounts of work, to determine kinematic and kinetostatic parameters achieved through graphic constructions. However, the accuracy of the results is not high due to errors of representation and measurements.

The analytical method is more accurate, kinematic and kinetostatic parameters are determined by solving corresponding equation systems for as many consecutive positions of the leading element. The workload is also high in this case due to the large number of systems of equation to solve, but this can be overcome by designing and writing algorithms in different specific programming environments such as Matlab or Mathcad.

A productive and accurate kinematic and kinetostatic analysis method is the use of specialized applications, or of CAD / CAE applications. In both situations, a first step is to geometrically model the mechanism, respectively to define the input parameters, following the actual run of the analysis to visualize and interpret the obtained results. [1].

2. Simulation Mechanical. Mechanical Event Simulation

The piston crank mechanism analysis was performed through the CAD / CAE Autodesk Simulation Mechanical, using the Mechanical Event Simulation concept (MES).

Autodesk Simulation Mechanical software, powered by Autodesk Nastran, provides a wide range of accurate mechanical simulation capabilities to predict product performance, optimize designs, and validate product behavior before manufacturing. Autodesk Simulation Mechanical enables users to utilize a highly accurate, industry-tested, general-purpose finite element solver to run comprehensive multiphysics simulations with other Autodesk products. Supporting multi-CAD environments, Simulation Mechanical provides flexibility in simulating CAD models in various formats. Integrated with the comprehensive Digital Prototyping solution offered by Autodesk, Simulation Mechanical brings finite element analysis (FEA) to all designers, engineers, and analysts to help them make great products [2].

Autodesk Simulation Mechanical is a stand-alone FEA mechanical simulation solution that provides a complete offering of capabilities, such as: a wide range of analysis capabilities, easy-to-use interface, robust solver technology, extensive material model library, multi-CAD modeling, Autodesk SimStudio Tools, multiphysics simulations, flexible cloud solving and Autodesk product integrations [2].

The Mechanical Event Simulation (MES) concept introduced and developed by ALGOR Inc.1 represents a paradigm shift in engineering design. It allows engineers and designers to simulate the actual conditions that a mechanical component will experience; that is, the event associated with its application. This is possible because MES accounts for both the interaction of the component with its surroundings, and the inertial forces generated by the motion of the component itself [3].

Event simulation, as an engineering methodology, is vastly different from the techniques that have been taught to engineers since the onset of formal engineering training begun by the Greek mathematician Archimedes around 200 BC. Event simulation is engineering by simulating a physical event in a virtual laboratory. To perform an engineering analysis using event simulation requires a different viewpoint from that of a classical stress analysis. Here we will not only define event simulation, but also contrast it with classical stress analysis [3], [4].

In most cases the finite element analysis is performed using linearity theory, but in many situations, for example, the motion of a mechanism is characterized by a non-linear effect [5].

The nonlinear analysis substantially differs from the linear analysis. The principle of superposition is not valid and only one load case can be treated at a time. Also, the loading history depends on the sequence of load application, as well as on the presence of initial stresses, such as residual stresses or prestressing [5].

MES combines the dynamics of rigid and flexible body with the possibilities of nonlinear stress analysis. As a result, MES can simultaneously analyze mechanical events involving large deformations, nonlinear material properties, kinematic motion and forces caused by that motion, and then predict the resulting stresses. Some of the main advantages of MES are the need to make fewer assumptions. With MES, there is no need for elaborate hand calculations, interpretation of results or experiments to determine the equivalent loading. The fewer assumptions that must be made reduce the chance of errors [6], [7].

By using Mechanical Event Simulation (MES) with Nonlinear Material Models, kinematic parameters, and stresses under dynamic load capabilities can be calculated. Loads can be constant or variable over time, calculations including inertial effects [2].

3. Achieving geometric model of piston crank mechanism

3.1. 3D model of the mechanism. Discretization of the assembly

The 3D model of the mechanism was developed in the Autodesk Inventor Professional CAD application, using the principle of parameterized modelling for each component, i.e. assembling them in an initial position of the mechanism by properly applying the assembly constraints, [8]. The 3D model of the mechanism is presented in Figure 1.

The 3D model of the mechanism, imported into the Simulation Mechanical application, was discretized with 50% fineness, using all tetrahedral elements instead of the default brick type, in order to avoid possible convergence errors during the analysis. The meshed model of mechanism can be viewed in Figure 2.

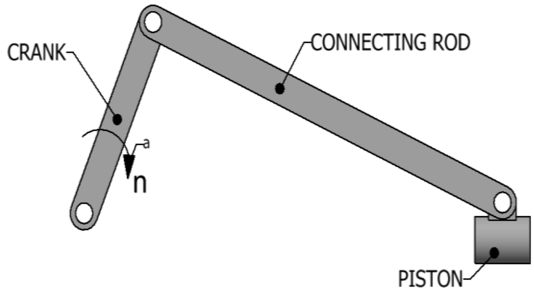

Figure 1. 3D model of the mechanism

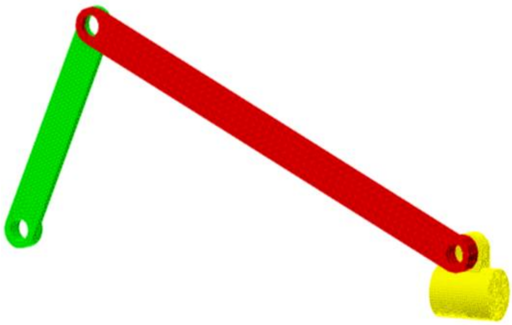

Figure 2. Meshed model

3.2. Modeling of kinematic couplings

The kinematic couplings are movable links between the components of the mechanism; they have been modeled as mesh joint in the appropriate boreholes of the kinematic elements. Figure 3 shows the kinematic coupling between the piston and the rod, and Figure 4 the one corresponding to the free end of the crank. The last coupling is actually defined between the crank and a fixed element, which, not being required for the analysis, is not represented.

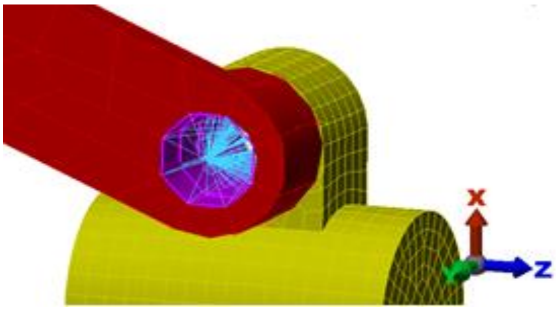

Figure 3. Pin Joint piston – rod

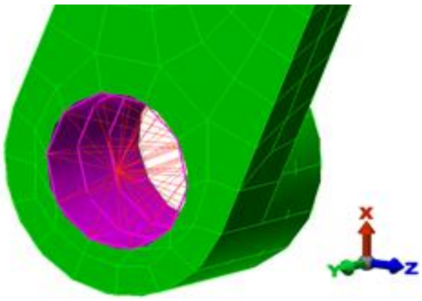

Figure 4. Pin Joint crank

An important aspect in modeling kinematic couplings is the correct definition of the element type. By default, the type of the element is truss type, but for the coupling of the free end of the crank a beam element was chosen, since a forced rotation motion has to be defined for this. Beam type elements have degrees of freedom of rotation, while the type truss, not [9].

3.3. Constraints. Required movement. External loads

Constraints are intended to limit the possibilities of motion (degrees of freedom) of elements to which they apply. Three constraints have been defined for the elements of the mechanism:

- The piston element performs only a translational motion along the Z axis, according to the axis system associated with the model, in this regard the other five possibilities of movement (three rotary and the other two translational) are constrained, according to Table 1.
- The movement of the mechanism is performed in the X-Z plane, according to the axis system associated with the model, in this regard the translation motion on the Y axis (perpendicular to the plane of motion) was constrained for all crank surfaces, considered the driving element according to Table 1.
- The crank performs only a rotary motion around the Y axis, according to the axis system associated with the model, in this regard, in the central node of the coupling from the free end of the crank, associated with it, all the five possibilities of motion have been constrained, as shown in Table 1.

Table 1. Surface and nodal general constraints

Surface/Vertices	Tx	Ty	Tz	Rx	Ry	Rz
Piston surface	Yes	Yes	No	Yes	Yes	Yes
Crank surfaces	No	Yes	No	No	No	No
Couple – crank node	Yes	Yes	Yes	Yes	No	Yes

The crank is the driving element of the mechanism and performs a complete rotation. Enforce of this motion is accomplished through a Nodal Prescribed Displacement.

Prescribed Displacement causes the node to rotate a specified quantity (Magnitude) around the Y axis, perpendicular to the plane of motion. In fact it has been created a new node, linked to the model by an elastic border element. In analysis by MES, for Prescribed Displacement to be used, a multiplier must be specified, whose value will be multiplied by the value Magnitude, the product representing the actual displacement applied to the new node [10]. In this regard, Prescribed Displacement will be made after a Load Curve, presented in Figure 5.

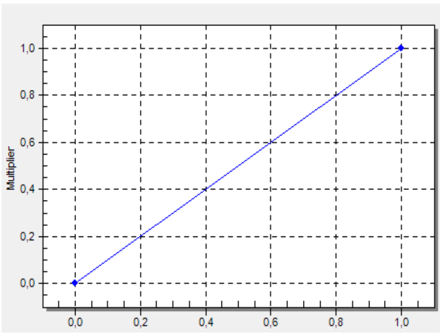

Figure 5. Nodal prescribed displacement Load Curve

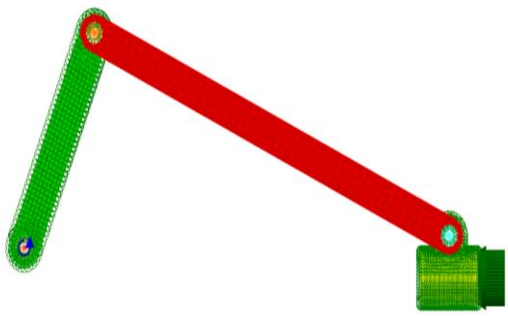

Figure 6. Mechanism model

External loads were defined by a force distributed to the front surface of the piston. The mechanism model, ready for analysis is shown in Figure 6.

4. Mechanism analysis. Visualization and interpretation of results

In order to perform the analysis of the gear unit, its parameters have been defined. Mechanical Event Simulation (MES) is defined by two parameters, the event length and the number of time steps. Event length is the time when the crank piston mechanism performs a kinematic cycle, or the crank performs a complete rotation and has been determined according to its speed. For the analysis, a ten-step count has been set; it will divide the duration of the event into equal time intervals, for which successive results have been obtained. Established analysis parameters can be seen in Table 2 [11].

Table 2. MES analysis parameters.

Duration (s)	Number of time steps
1	10

After performing the analysis we obtained results of the stress type von Mises, the displacement, strain and safety factor for each step of the kinematic cycle. For example, Figures 7 and 8 show the state of stress for the entire mechanism at time steps 2 and 7.

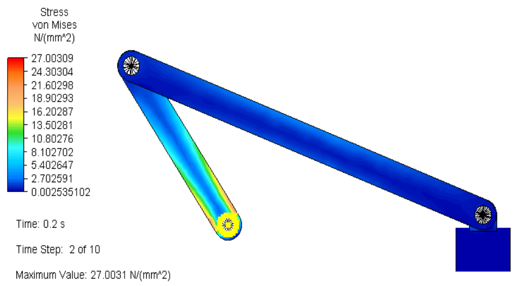

Figure 7. Stress state von Mises at time step 2

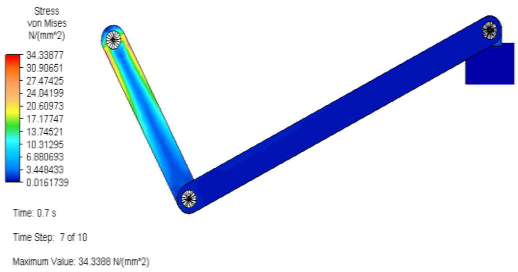

Figure 8. Stress state von Mises at time step 7

Figures 9-14 show von Mises stress state for each element, indicating the node in which the stress is maximum, respectively variations of these stresses over a kinematic cycle.

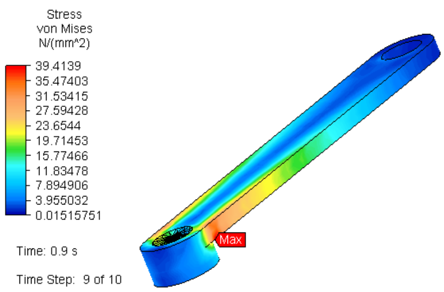

Figure 9. Crank stress state

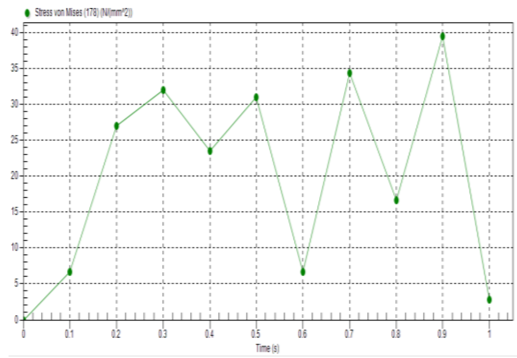

Figure 10. Maximum crank stress variation according to time

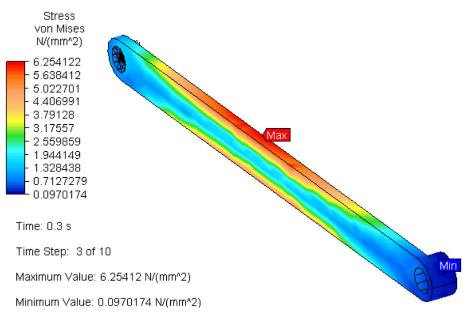

Figure 11. Rod stress state

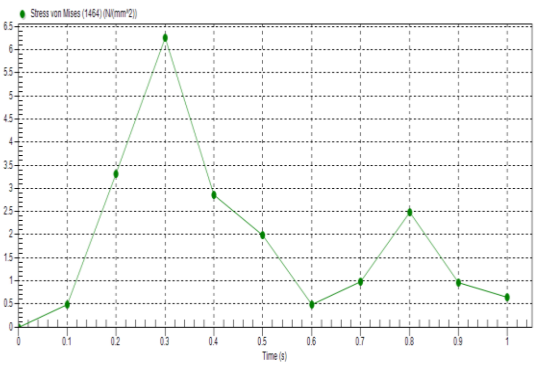

Figure 12. Maximum stress variation for rod in time

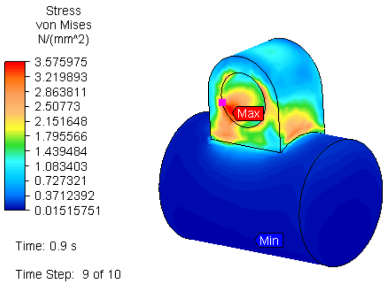

Figure 13. Stress state for the piston

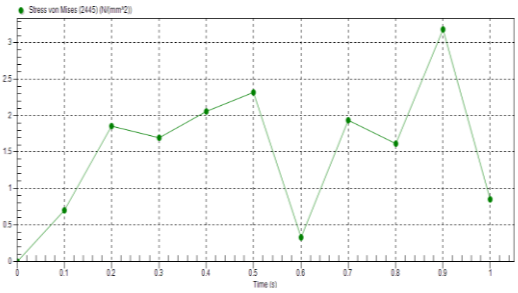

Figure 14. Maximum stress variation for piston in time

Figures 15 and 16 show the state of the reaction force in the coupling at the free end of the crank, relative to its central node, i.e. its variation over a kinematic cycle.

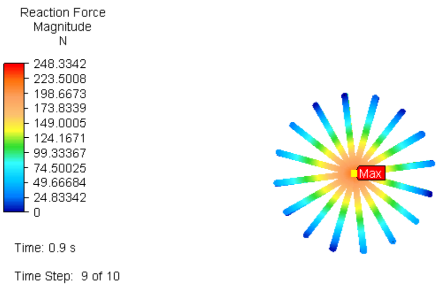

Figure 15. Motor coupling reaction force

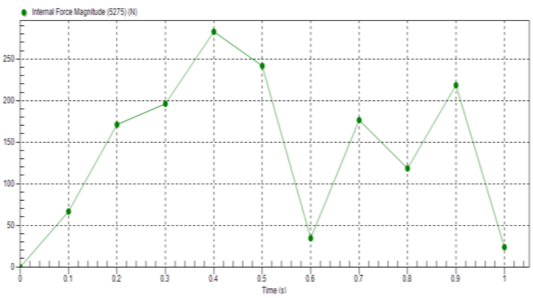

Figure 16. Reaction force variation from motor couple in time

Analyzing the results presented in Figures 7 - 16, we can specify the following issues:

- After the analysis, we can observe a similar stress state for each time step of the kinematic cycle, at each moment the crank being the most stressed kinematic element;
- Maximum crank stress occurs at time step 9 in node shown in Figure 9;
- Maximum rod stress occurs at time step 3 in node shown in Figure 11;
- Maximum piston stress occurs at time step 9 at the node shown in Figure 13;
- Maximum value of the reaction force at the free end of the crank occurs at time step 4, in the central node of it.

5. Conclusion

Based on the above, the finite element analysis of the crank piston mechanism can be considered a useful method when designing them. The use of MES with the nonlinear material model, by properly establishing the analysis parameters, allowed the results to be obtained at different time intervals of the event. In this paper, the obtained results showed the distribution of the maximum values of stresses, respectively their variation over a kinematic cycle for each element.

References

[1] Manolescu N, Kovacs Fr and Orănescu A 1972 *Teoria mecanismelor și a mașinilor*, Editura Didactică şi Pedagogică, Bucureşti, România
[2] *** https://www.cadac.com/media/2000/simulation-mechanical-2016-brochure.pdf
[3] Giosan I 2006, Dynamic analysis with damping for free standing structures using mechanical event simulation, *U.P.B. Scientific Bulletin* **68**(3) 15 - 30

[4] *** http: //download. autodesk.com/us/algor/userguides
[5] Spyrakos C C and Raftoyiannis J 1997 *Finite element analysis in engineering practice,* Algor, Inc. Publishing Division, Pittsburg, PA, USA
[6] Miklos I Z, Miklos C and Alic C I 2018 Finite element analysis of cylindrical gear mechanical event simulation, *IOP Conf. Ser.: Mater. Sci. Eng.* **393** 012046
[7] ***http://www. knowledge.autodesk.com/support
[8] Dolga L, Revencu M, Maci C A and Giuchici M 2004 *Parametric and feature based modelling with applications in Catia and Inventor*, Politehnica Timișoara Publishing, România
[9] Nam – Ho K and Bhavani V S 2009 *Finite element analysis and design,* John Wiley & Sons Inc., USA
[10] ***http://help.autodesk.com/ view/ ASMECH/2017
[11] Ranjith raj A, Udhayan V and Veerappan M 2017 Transient analysis on a four cylinder IC engine using ANSYS, *International Journal of Advanced Research in Management, Architecture, Technology and Engineering* **3**(2) 184 - 187