ARTICLE

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)

مـــجلـة جـــامعة بـــابـل للعلـــوم الهندسية

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

The full Design and Build of Pressure Switch from available cheap Materials using Finite Element Analysis of some of its non- metallic Parts working synchronously inside the Pressure Switch

Ahmed H. Al-barban

Department of Ceramic and Construction Materials, College of Material Engineering University of Babylon, Babylon, Iraq

Mat.ahmed.hamad@uobabylon.edu.iq

Received:	28/5/2025	Accepted:	17/7/2025	Published:	28/8/2025

Abstract: -

Waste PVC parts and waste rubber materials are used to improvise a pressure switch. Also, metallic parts are involved in building that pressure switch. The design of pressure switch considers the using of cheap waste materials and involves so simple design that any primitive technic can build this item. Furthermore, the design of pressure switch puts into considerations the hazard associated with rubber's diaphragm leakage or failure. The dimensions of switch's parts are selected based on computation using finite element analysis for the two pieces of rubber which are synchronously working. The first rubber piece works as a diaphragm while the second, which is strip, works as spring as well as adjustable tensioner to regulate the set value of pressure. There are two metallic parts working on transferring the motion from the diaphragm to strip. The first metallic part is spindle and the second is the leverage which is connected to the electrical contact too. The two rubber pieces and the two metallic parts are drawn in Catia R17 software, then they are imported to Ansys Workbench 2022 software where a three dimensional modeling is created. The purpose of this modeling is to find out the relation between the applied pressure on the diaphragm and the resulting deformation on the strip which is previously stretched to corresponding certain value of pressure. Depending on the results obtained from the modeling, a real pressure switch is built with the same dimensions in modeling. Then, that real pressure switch is tested using the same input pressure values in the modeling. The practical deformation values are compared to the corresponding values obtained from the Ansys Workbench 2022 modeling. The comparison shows that the experimental deformations are higher at lower pressure values because the mechanical properties of the old real rubber are degraded with time while at higher pressure vales the real deformations become less than those in the modeling because the friction in real joints increases with high pressure and also, in Ansys Workbench 2022 modeling, the friction in joints is neglected. Anyway, there is little difference in magnitudes of deformation but generally both practical and modeling results have the same behavior and manor besides they follow the same logical path.

Keywords: Stress, Strain, Rubber curve fitting, Adaptive Remeshing, Deformation, Friction, pressure, Augmented Lagrange, Solution Convergence

ARTICLE
JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلـة جـــامعة بـــابـل للعلـــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

<u>Introduction:</u>

Pressure switch is a vital tool used in industrial applications like well pumps, furnace, HVAC systems…. Etc[1]. Pressure switch works on principle of closing electrical circuit when pressure reaches a specified amount which is previously set [2]. There are many types of pressure switch, depending on design and application, like diaphragm, bellows, bourdon-tube, and piston" [3]. In this paper, a diaphragm type is dealt with and formed from an old tube's rubber sheet. In the diaphragm type design, the deformation of the diaphragm resulting in fluid's pressure is transferred to open and close the electric contact [4]. Therefore, the diaphragm is in contact with other devices to transfer the movement to sensing parts like sensor or electric contact item [5]. So this diaphragm should be made from flexible material like rubber and works as membrane too to separate two different pressure chambers. The rubber diaphragm has simple design and can be used in many applications besides being cheap [6].

<u>Improvisation of the pressure switch:</u>

The pressure switch in this paper is super simple and is improvised from cheap available materials which are found as waste materials. The parts of the improvised switch are:

1. Union fitting
2. Frame
3. Spindle
4. leverage
5. PVC cap

1. <u>Union Fitting:</u>

This part is the heart of the item where the rubber diaphragm is contained. The rubber diaphragm is rubber rig cut from an old tire's tube in circular form with a diameter of 25 mm. The union fitting used in this paper is an ironic one with 0.5 inch size and it consists of two main subparts besides ring to connect the two main subparts. The diaphragm is placed between the two main subparts and the whole assembly (subparts and diaphragm) is connected together by the ring.

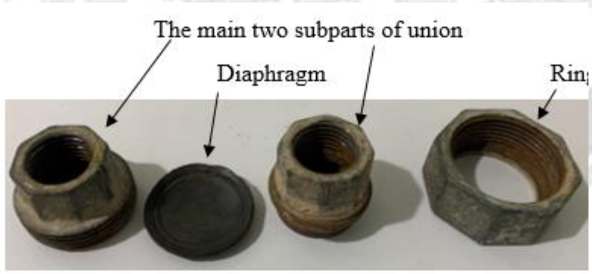


Figure 1 Real union fitting parts with the diaphragm

ARTICLE
JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلــة جـــامعة بـــــابـل للعلـــــوم الهندسية
Vol. 33, No.4. \ 2025
ISSN: 2616 - 9916

2. Frame:

is a combination of two T shape PVC fittings, and three small PVC pipes. Also, there is long PCV pipe which extends vertically in frame to contained the spindle as will be shown later. All these subparts are connected as shown in Figure 2 below to form the whole frame.

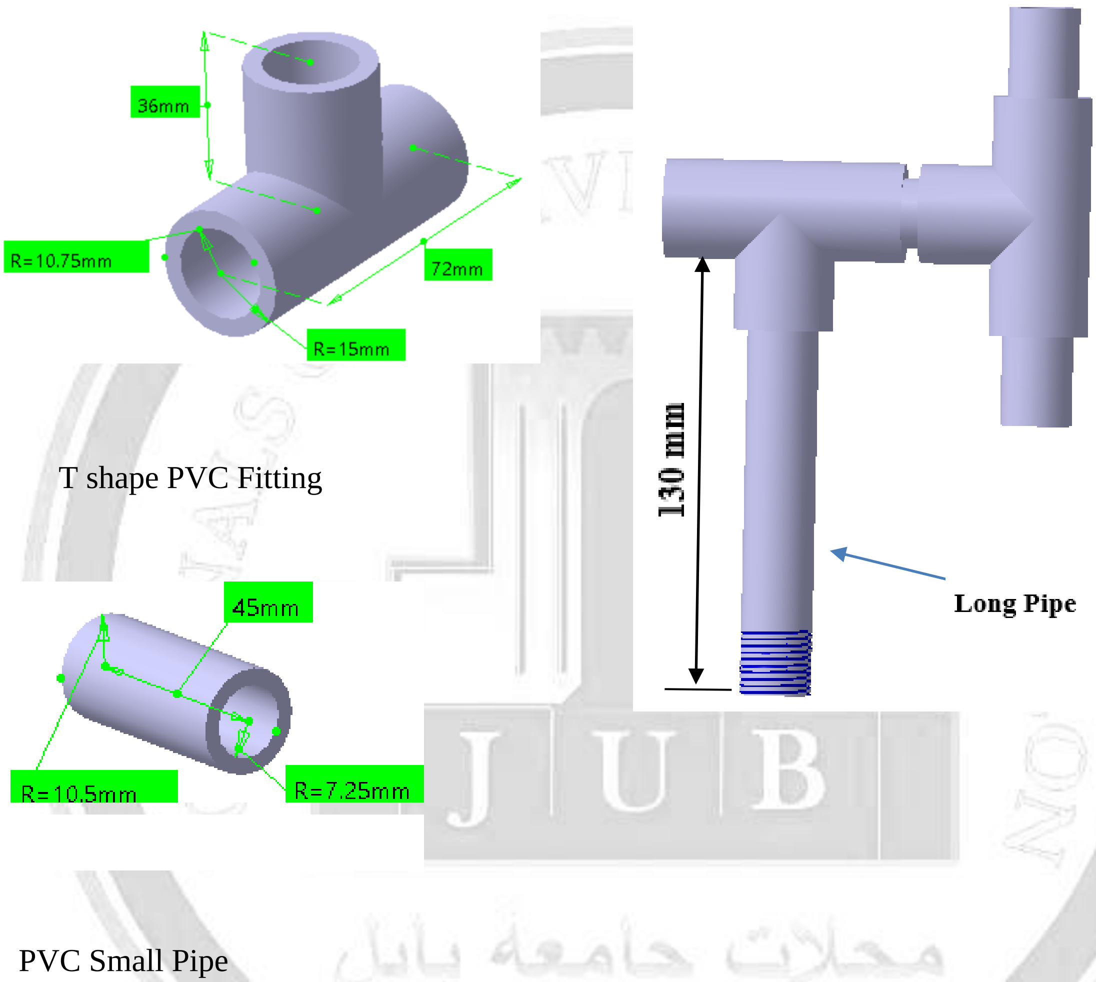


Figure 2 Frame of the pressure switch with its parts

Note: the long pipe has a thread, made by 0.5-inch dice, at the bottom in order to be tighten in with union fitting. This thread can be used for adjustment of pressure set point.

3. Spindle:

This part is about 9 mm diameter threaded rod with a length of about 155 mm and it is found in bicycle wheel for connecting the wheel to bicycle's frame. Also, this part can be made from 9 mm threaded long bar. The spindle erects vertically inside the long pipe in frame of pressure switch. There is small nut threaded to the spindle at the bottom to be in touch with diaphragm as will be shown later. A hole of 4 mm is drilled at the top of the spindle to be engaged with leverage where the motion of the spindle is transferred to the leverage. There are two extra nuts threaded to the spindle with same outside diameter as inside diameter of the long pipe. These two nuts work as a guide to the spindle for moving upward and downward without any swaying. Also, those nuts work as obstacles for liquid or water to flow up in case of leaking in the diaphragm. The spindle part is shown in

Figure **3** below

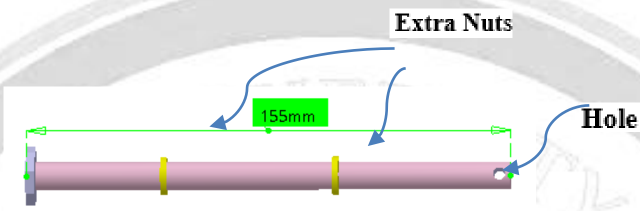


Figure 3 Spindle part (drawn in Catia R17)

4. Leverage:

This part, leverage, is formed from 30 cm length grilled rod (skewer)as shown in Figure 4 below, the skewer has a diameter of 3 mm and is formed to improvise the leverage in such a way that its circular edge is removed and two right angle bends are made as shown in Figure 5 below. the task of the leverage is to transfer the motion from the spindle to the electric contact (limit switch).

Figure 4 grilled rod (skewer)

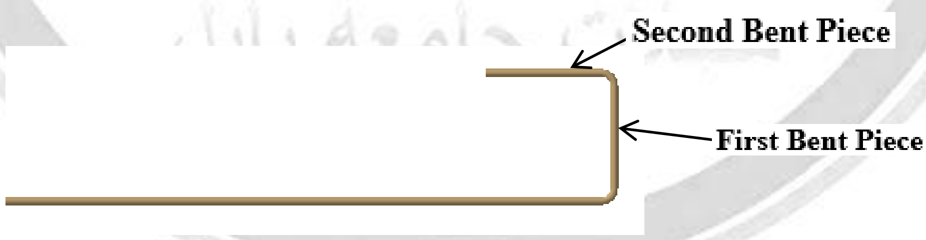


Figure 5 the leverage

5. PVC cap:

It is normal bottle’s cap used, in daily life, to seal the bottle of beverage. But in this project, it is used to support the leverage as shown in Figure 6 below. The cap is inserted to the T fitting after subjecting a heat because both T fitting and cap have same external diameter. Also, a hole is drilled in the cap to let the leverage be inserted through. The position of that hole is selected based on motion maneuver of the leverage inside switch and strength of the cap to loads i.e. the position of this hole is not roughly selected but based on results obtained from stress analysis associated with preload resulting in inserting the cap into the T fitting, as will be seen later in this paper.

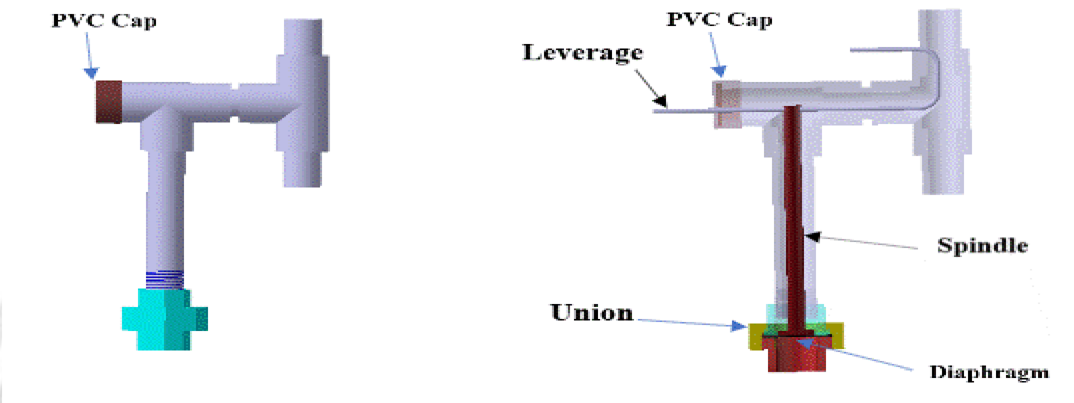


Figure 6 The cap inserted to T fitting

Working Principle:

The fluid, generally water, will apply pressure on diaphragm and the latter will be inflated out. This inflation is proportional to the pressure’s amount and will push the spindle upward moving the leverage upward too. The second bent end of the leverage is connected to the electric contact (limit switch) which will close the electric circuit. When the pressure drops on the diaphragm, the spindle will move back downward to a certain position, depending on the amount of the new pressure, leading the leverage to move downwards and causing the electric contact to be opened at the end.

The explained design above is not practical enough because it does not work with high pressure’s values and it works only on specific amount of pressure i.e. it cannot be set to work on range values of pressures. Also, when pressure drops, the movement of spindle downward depends on gravity, and this gravity might not be enough or might be lower than other hampering forces, like friction, causing the electric switch to be stuck closed.

Therefore, a strip of rubber is added to the design, this strip is connected to leverage on one side and to the frame on another side as shown in Figure 7 below. This rubber strip can be stretched down, from its lower side, to certain amount so the tensile in this strip can be set accordingly. This pretension in the strip will restrict the motion of the leverage upward therefore, higher pressure’s value will be required to move the leverage up. Based on that, the stretching settings will be an indication of pressure settings in the switch. Also, this pretension will guarantee the leverage and spindle to move downward as soon as pressure drops.

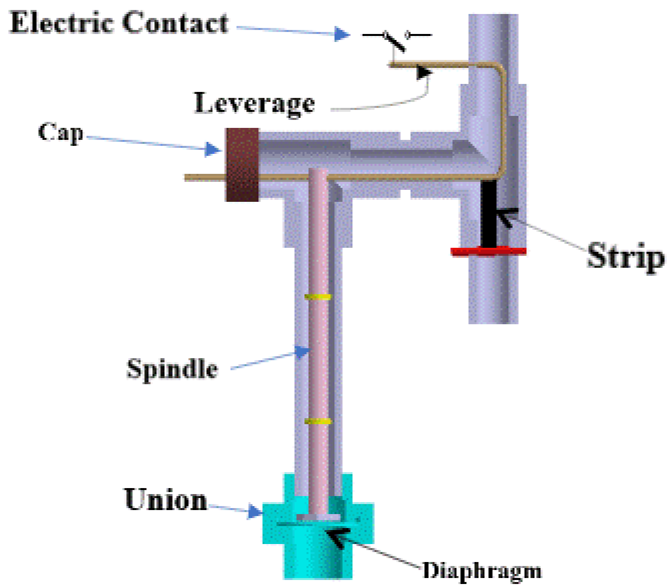


Figure 7 Strip in Pressure Switch

Based on the explanation above, both a diaphragm and a strip will work synchronously therefore, a certain simulation is required to optimize the size of the strip and the range of stretching.

It can be guessed it will be easier and more simple if the pressure switch is designed in such a way that spindle is connected directly to electric contact without the leverage and all its associated frame's parts. But that simple design is dangerous because any water leak from diaphragm will reach the electric contact (limit switch) causing big hazards. Therefore, there is a need to prolong the path of motion and doing some modifications in this path to prevent any droplet of fluid from reaching up to the electric contact. The modifications which prevent the droplets from reaching upward are

1. Drilling several holes in the long pipe of frame to let water flows out, in case of diaphragm leaking
2. Amounting two extra thin nuts on the spindle to restrict the fluid from flowing upward and let it (fluid) flow out through the holes previously drilled in the long pipe.
3. Placing the second T fitting in this position will let droplet of water flow outside and preventing flowing upward.

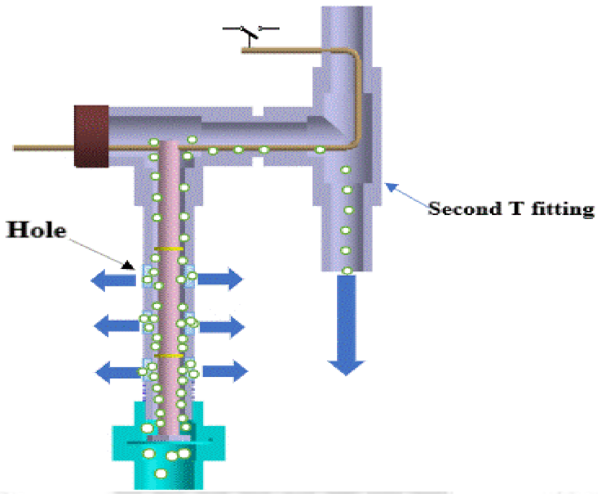


Figure 8 Fluid droplets coming out of device in case of diaphragm leak

Note: the right hand side of the leverage is moving upward and downward within groove in pipe which works like guide. As will be seen later in practical part of this paper.

Calculations and Simulations:

The dimensions of this pressure switch's parts and its design are not arbitrary improvised but are based on optimization and multi simulations. Before dealing with simulation of the rubbers parts of the switch, there is a need to make simulation to the cap to see if the region on which the hole is drilled is safe and do not form stress concentration. So, it requires modeling analysis using Ansys 2022 Workbench software.

Simulation of the cap:

The insertion of the cap into T fitting will generate stresses in the cap. So, drilling any hole on the cap might result in forward consequences like tearing or cracking. Therefore, the place where drilling is should have minimum stresses and does not cause any stress concentration [7]. The dimensions of the cap are measured and the cap is drawn by Catia V5R17 as shown in Figure 9 below, knowing that the thickness of cap is about 1 mm. Then it is imported to ANSYS Workbench 2022 where three dimensional modeling is created. The external cylindrical face is divided to form edge at which a certain boundary condition is applied as will be seen later [8].

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

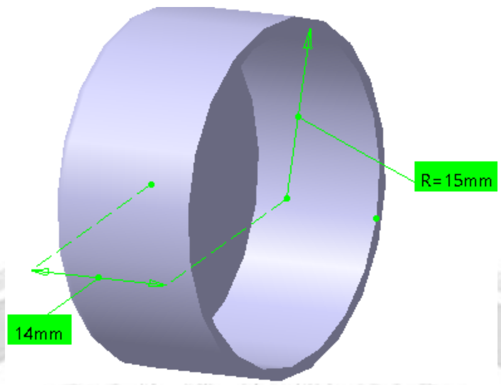


Figure 9 Cap drawn in Catia V5R17

Meshing:

a hexahedron element type is selected for meshing, with 1 mm element size, to get a faster solution with acceptable accuracy [7]. The total number of element used in this simulation is 9477 with total node number of 36690. Also, several mesh option is included like mapping some faces, the mesh is shown in Figure 10 below.

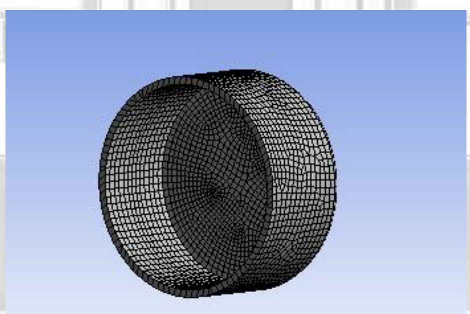


Figure 10 Meshing of Cap

Boundary condition:

The first boundary condition is a displacement applied at the front face of the cap as shown in

Figure **11** below. This boundary condition makes the face has zero displacement in longitudinal direction(Z) while the other directions are set to be free. This boundary condition guarantees free planar motion without any motion in longitudinal direction.

ARTICLE
JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلــة جـــامعة بـــــابـل للعلــــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

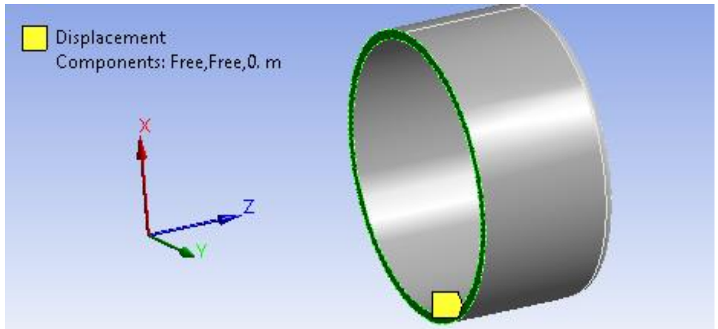


Figure 11 First boundary condition

Second boundary condition is applied at the edge where external cylindrical face is divided. A remote displacement is applied where a zero displacement in x direction is set to be zero while the other displacements in other directions are set to be free including all rotational displacement. The second boundary conditions are shown in Figure 12 below

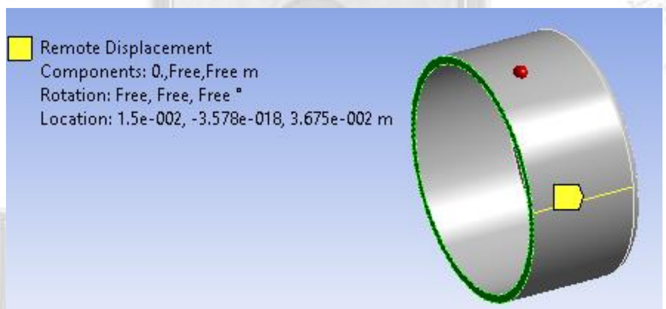


Figure 12 Second boundary condition

Then a pressure of 100 Pa is applied to the internal cylindrical face of the cap as shown in Figure 13 below. This amount of pressure is arbitrary selected because the purpose of the load is not to find stresses amount but to find stress distribution and concentration.

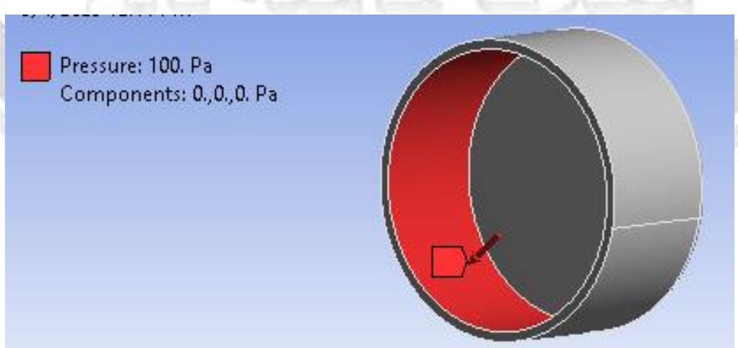


Figure 13 Applying pressure

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

Then, solution is done, and the results are obtained as shown in Figure 14 below.

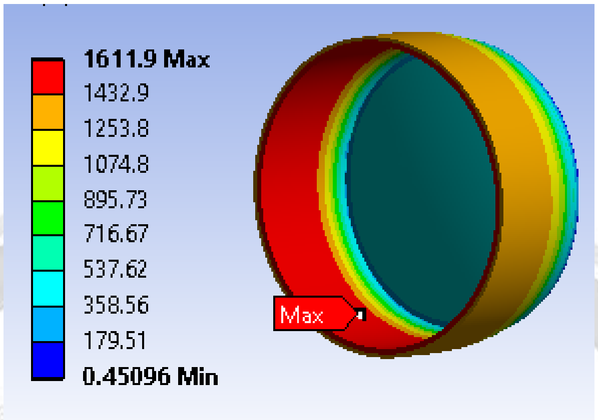


Figure 14 Stress distribution

From the results shown in figure 14 above, the maximum stresses are in the cylindrical faces of the cap while the lateral face, where a hole is about to be drilled, has the minimum stress. So, any position on the lateral face can be selected to drill a hole.

Based on the results shown above, the 3mm diameter hole is drilled in the cap as shown in Figure 15 below. This diameter is the same as leverage diameter and this position of the hole will award the other side of leverage a higher maneuver to move upward and downward inside the frame.

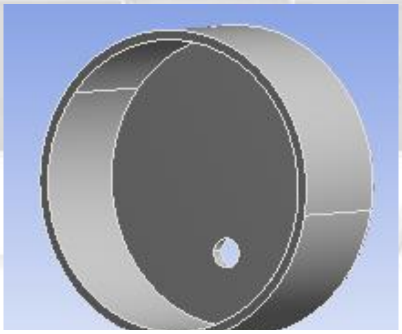


Figure 15 a drilled hole position

To be sure, the new cap with a hole is safe to be used, a new modeling is done using Workbench 2022 with the same load and boundary conditions as with former simulation. The mesh is done as shown in Figure 16 with 1 mm element size. The total number of element used in this simulation is 6293 with total node's number of 25511.

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

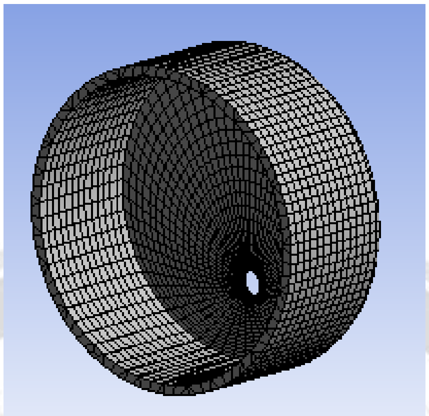


Figure 16 meshing of the cap with hole

The solution is done, and the results are obtained as shown in Figure 17 below

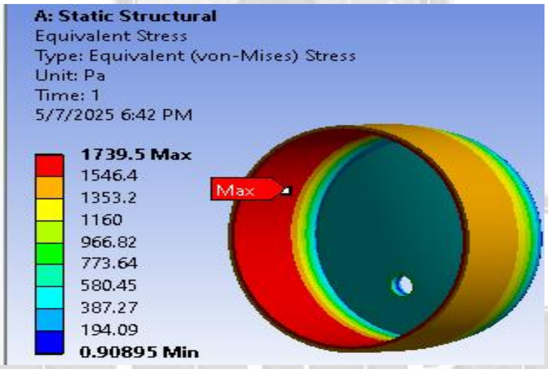


Figure 17 Stress distribution on cap with a hole

The results above show there no issue of the selective position. Also, this hole's position makes the leverage move in maximum working space through the frame. This hole will work like universal joint for leverage to let the other side of the leverage move freely upward and downward.

Simulation of the Rubber Parts:

Geometry:

As was explained earlier in this paper that the diaphragm and strip work synchronously, so the strip and the diaphragm should have synchronous simulation too i.e. it can neither be taken simulation of diaphragm alone nor strip alone. Therefore, a three dimensional modeling is built for simulation. There is no need to import all parts of the switch for simulation but just the parts which are connecting the two rubber pieces i.e. spindle and leverage besides the two rubbers (diaphragm and strip) are the only involved parts in the simulation while the frame and union are neglected. The geometries of leverage and spindle, used in simulation, are simplified and modified to be suitable for applying the boundary conditions. Figure 18 below shows the modified geometry in Ansys Workbench 2022. it can be seen that the leverage is modified a lot to facilitate applying the boundary condition. Also, the frame and union fitting are neglected.

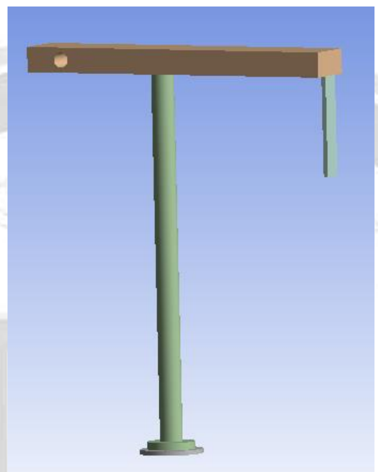


Figure 18 geometry of pressure switch in Workbench 2022

Material property:

In this simulation, the material property is defined for just diaphragm and strip because the other (leverage and spindle) are defined as Rigid in the modeling. Generally, rubber is highly deformed under small applied load and it retain to its original shape after load's removal [9]. Also, rubber is well known to be a highly nonlinear material i.e. stress- strain diagram does not follow linear relation [10]. so, there is a need to use constitutive modeling for analysis this relation, like Rivlin model,neo-Hookean model, Mooney- Rivilin model, Yeoh model, Ogden model…etc[11]. The rubber material in this modeling is based on Elastomer Sample (Yeoh) depending on former results obtained by former study publish in paper [12] which show that the old tube's rubber material's analysis approaches that of Elastomer Sample (Yeoh). Therefore, the tests of the uniaxial, biaxial, and shearing data of this Elastomer need to be fitted and the resulting corresponding curves should not cross each other and can approach tests' points with minimum error [9]. So, the best choice for this fitting is by using Yeoh-3rd order material modeling based on absolute error for fitting. The strain energy function for the Yeoh-3rd order hyperplastic model is [13]:

$$\psi = C_{10}(\overline{I_1} - 3) + C_{20}(\overline{I_1} - 3)^2 + C_{30}(\overline{I_1} - 3)^3 + \frac{1}{d_1}(J-1)^2 + \frac{1}{d_2}(J-1)^4 + \frac{1}{d_3}(J-1)^6$$

Where: -

C_{10}, C_{20}, C_{30} - material constants

ARTICLE
JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلـة جـــامعة بـــــابـل للعلـــــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

d_1, d_2, d_3 - incompressibility parameters

$\bar{I}_1$- - the deviatoric first principal invariant

J - the Jacobian

After solving the material modeling with Workbench 2022 software, the values of the material constants and incompressibility parameters were computed. Figure 19 below shows the resulting fitting curve.

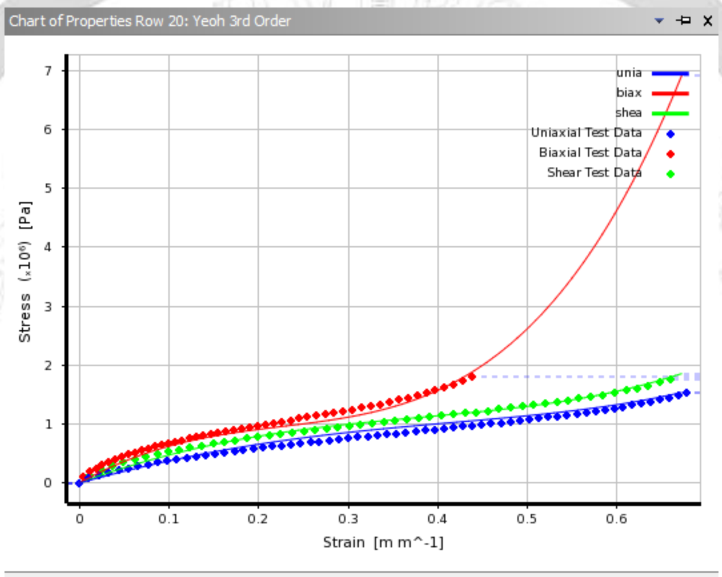
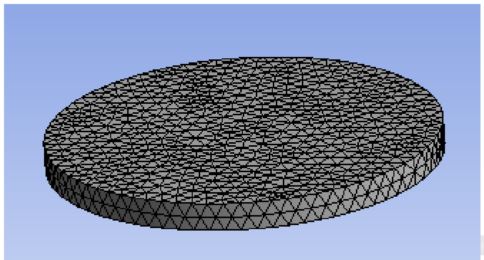
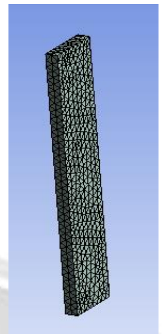


Figure 19 Curve Fitting based Yeoh 3rd Modeling

Meshing:

Both diaphragm and strip subject to high deformation and there might be element distortion within loading or solution [14]. So, some details are required to be done in mesh setting, like the physical performance is changed from mechanical (default) to nonlinear mechanical and element order is changed to linear. Also, an adaptive meshing system is activated in such a way that the software makes Remeshing within solution to avoid high distortion in elements. Remeshing of the model within solution is based on skewness value of 0.6 to detect minimum distortion in elements [15]. The element type used in this modeling is tetrahedron, as recommended by adaptive meshing option, with element size of 0.0001 m, which is relatively small to avoid element's distortion. The total number of element used in the modeling is 12817. Figure 20 below shows the meshing of the diaphragm and the strip.





Diaphragm meshing Strip meshing

Figure 20 Meshing of Diaphragm and Strip

Boundary condition:

1. There is a need to find out the coefficient of friction between the bottom surface of the nut, connected to the spindle, and top surface of diaphragm in order to be used in modeling. so, to measure this parameter, the nut is placed on piece of rubber as shown in Figure 21 below. That rubber is the old truck tube's rubber. Then, the piece of rubber is gradually and slowly inclined until the nut is just about to slide down as shown in Figure 22 below. At this moment, the tangent of inclined angle represents the coefficient of friction [16]. This angle is about 35^0 for our experiment and the tangent of this angle is about 0.7

Figure 21 Experiment details to compute the coefficient of Friction

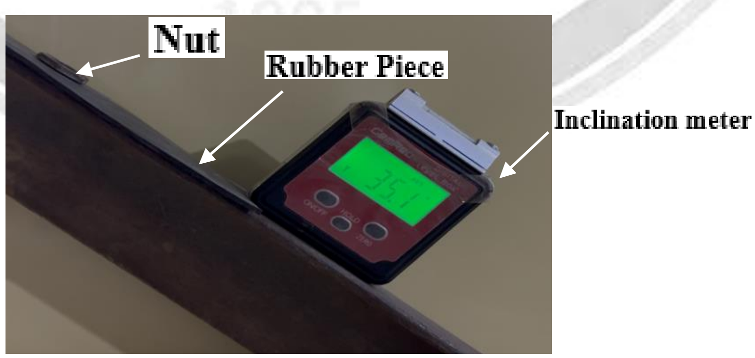


Figure 22 Angle of inclination

2. For the diaphragm, a fixed boundary condition is applied to the circumferential area as shown in Figure 23 below, then a pressure of 350 KPa is applied to the bottom surface of the diaphragm as shown in Figure 24 below. This pressure represents the fluid's or water's pressure, when 1 HP water pump is used, and it is applied gradually within 10 seconds in the program as shown in Figure 25 below. Also, in each second, there are other 10 substeps loading to avoid elements' distortion and divergence within solution. Because the diaphragm is in contact with spindle, a frictional contact is defined between the upper surface of the diaphragm and lower part of the nut which is threaded to the spindle. a coefficient of friction is set to be 0.7 based on practical experiment explained previously. Figure 26 shows the contact pair between the diaphragm and the spindle.

Figure 23 Fixed Support on diaphragm

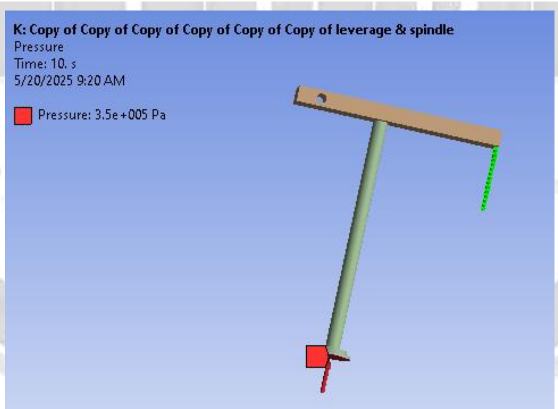


Figure 24 Pressure applied on diaphragm

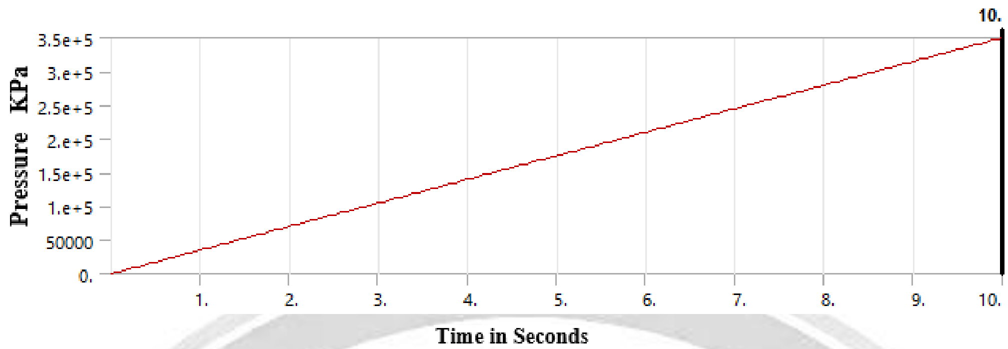


Figure 25 Time vs applied pressure

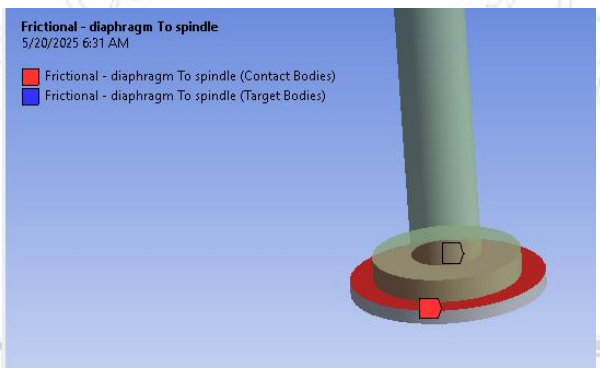


Figure 26 Contact Pair

3. For the spindle part, the spindle in the modeling is set to be rigid because its deformation is negligible as a compare to the rubber. Because it stands vertically and moves vertically too, translational joint is defined between the spindle and the ground. The ground is like the frame in reality. This translation joint will allow the spindle to move upward and downward only as shown in Figure 27 below

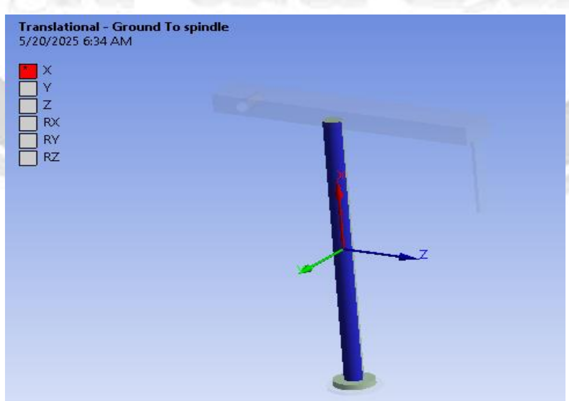


Figure 27 Translational Joint on Spindle

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

4. For the leverage, as was explained before, this part is modified in Software and is set to be rigid because its deformation is so small as a compare to the rubber. A fixed joint is defined between the right hand side face of the leverage and upper face of the strip, as shown in Figure 28 below, to represent the position where leverage and strip are connected. Also a rotational joint is defined between the inner hole surface of the leverage and the ground as shown in Figure 29 below. This joint represents the joint created with the cap in reality.

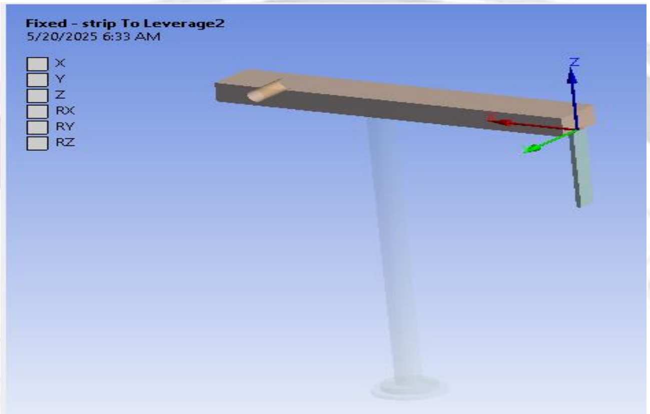


Figure 28 Fixed Joint

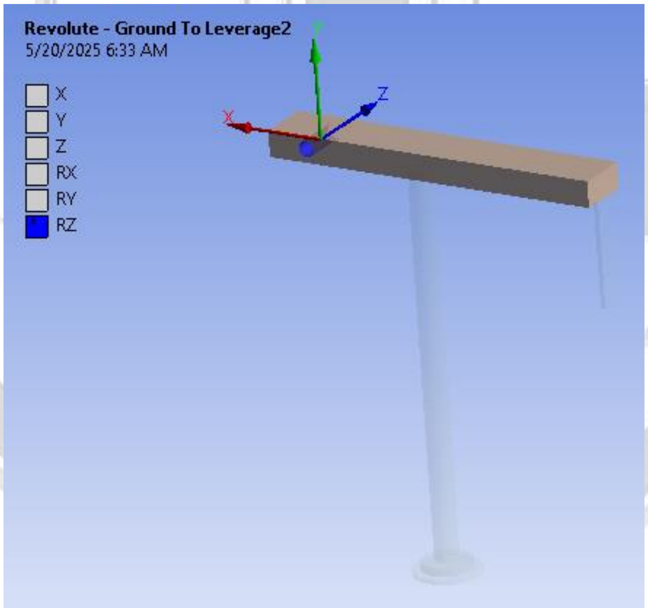


Figure 29 Revolute Joint

5. For the strip, a displacement boundary condition is defined at the lower surface of the strip as shown in Figure 30 below. This displacement represents the strip's stretch that it is previously set. In this modeling, the displacement is set to be 0.015 m and it is directly applied down without any gradual steps because the strip in reality is already stretched before the influence of pressure.

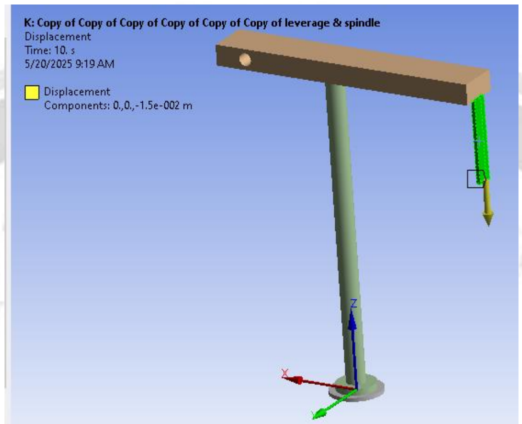


Figure 30 Displacement applied to the strip

6. Finally, a general joint is defined between the lower surface of leverage and upper surface of the spindle, this joint allows the perpendicular rotation between the two parts and allows displacement along the leverage i.e. the spindle can slide along the leverage. This joint represents the joint where the leverage goes through a hole in spindle. Figure 31 below shows the general joint modeling in Workbench 2022

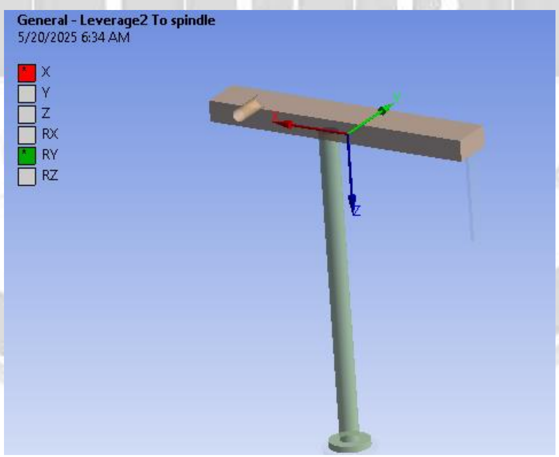


Figure 31 General Joint

Solution and Results: -

The modeling of rubber is already nonlinear and here, two rubber parts are synchronously working besides there is frictional contact, which is nonlinear too[17]. So, the solution will be highly nonlinear solution and it is very time consuming to reach the convergence. An Intel core i7 with 2.77 GHz computer is used for this simulation. Also, some changes are required to be done in analysis setting detail, like the auto time stepping is set to be On, initial substeps is set to 40, minimum substep is 10 and the maximum is set to 1000. The solution took long time until reaching the convergence and the results were obtained at the end. Figure 32 below shows the vertical deformation in the model, the maximum vertical deformation is in the strip with an amount of about 0.01 m at the top of strip or at the place where it is in contact with leverage. This deformation is the same the displacement of leverage touching the electrical contact item (limit switch). Figure 33 below shows the relation between the applied pressure on diaphragm and the resulting deformation in strip. Also, Figure 34 below shows that maximum von messes stress is in diaphragm with an amount of 3.4 MPa. The values of maximum deformation and stress occurs when the strip was previously stretched downward to 0.015 m displacement. Those values of deformation and stress might change if the strip's stretch value changes.

Figure 32 the Deformation's Results

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلـة جـــامعة بـــــابـل للعلـــــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

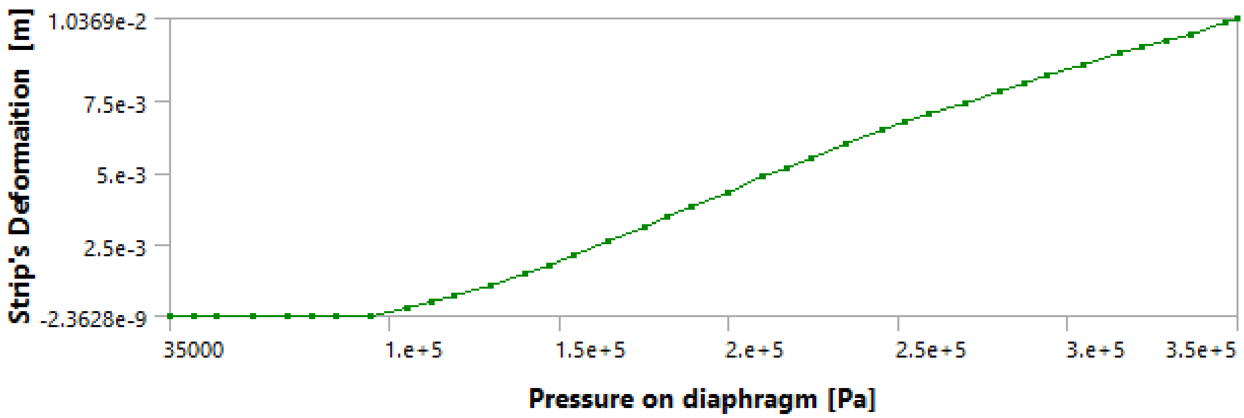


Figure 33 Applied Pressure vs deformation in strip

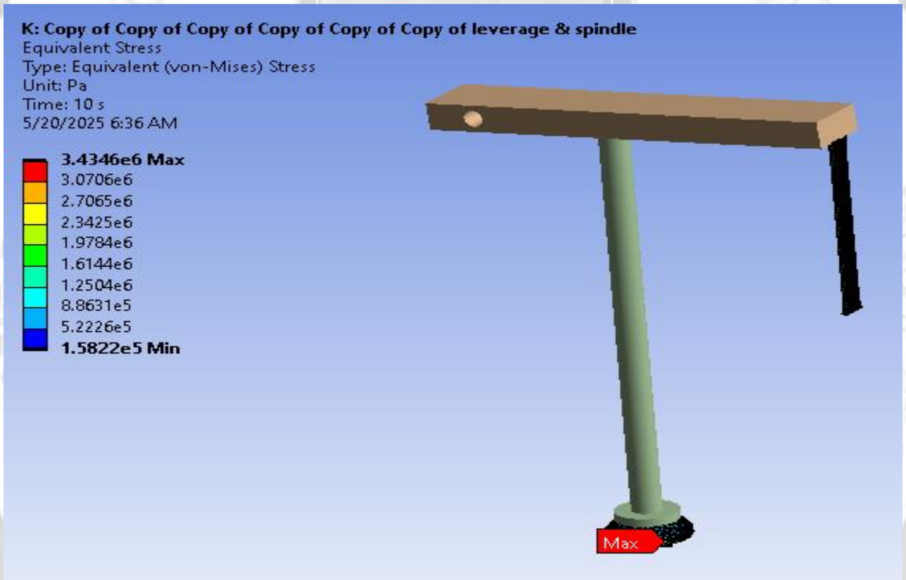


Figure 34 Stress' Results

Practical Work:

Depending on the results obtained from the modeling in Workbench above, the selective size of the PVC pipes of frame will be enough for 10 mm motion of leverage. Therefore, the dimension of the strip above will be dependent in practical work. The strip is cut from the same old tube in which the diaphragm is cut. Then, the strip is superglued to the leverage from the top and to the small bolt-nut at the bottom as shown in Figure 35 below. The bolt-nut will be used as adjustment for stretch of strip.

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلــة جـــامعة بــــابـل للعلــــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

Figure 35 Leverage with Strip

The frame is made from an old T shape pvc fitting after cleaning it using drilling bit to remove the remnant of the old pipe previously inserted. Then, frame is assembled from T shape fittings and PVC pipes, which are old too, as shown in Figure 36 below. It can be noted there is groove on the second T fitting to let the second bent of the leverage slide through.

Figure 36 the whole frame

Then, the union fitting with a diaphragm, the real spindle shown in Figure 37A ***Real spindle*** below, and the leverage are assembled all together to form the whole pressure switch as shown in Figure 37B below

Figure 37A Real spindle

Figure 37B Real improvised Pressure Switch

Figure 37 the real improvised switch with real spindle

<u>Note:</u>

In real spindle shown above, the two extra nuts have two task: to guiding the spindle inside the long pipe and to hamper water from flowing upward. Therefore, the two nuts are a little grinded to be fitted and to move freely inside the long pipe. Also, it can be seen there are holes in long pipe to let leaked water to getting out.

The test of the improvised switch:

To test the switch, it should be connected according to the diagram shown in Figure 38 below where water' pressure can be regulated by the tap. When zero pressure is required, the tap should be fully opened. But, when a certain value pressure is required, the tap should be gradually and slowly moved back toward the close position until the required pressure is read in the gauge mounted in the same pipe line feeding the pressure switch. Then, the displacement of leverage, which is the second bent of leverage, is measured by using Vernier caliper

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلـة جـــامعة بـــــابـل للعلـــــوم الهندسية
Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

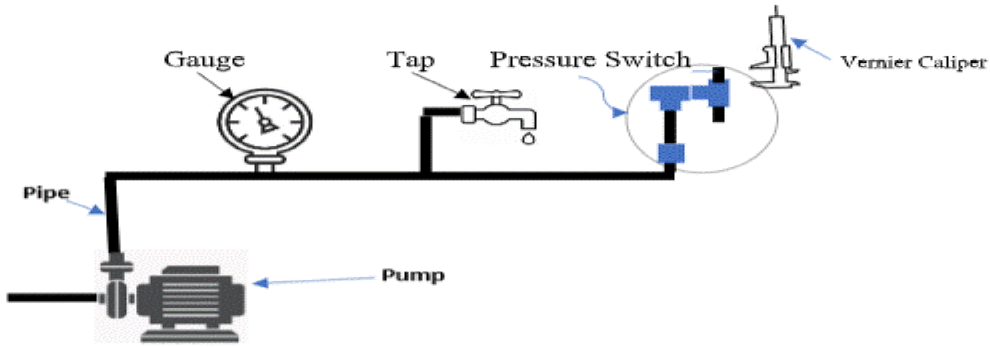


Figure 38 diagram to test the improvised pressure switch

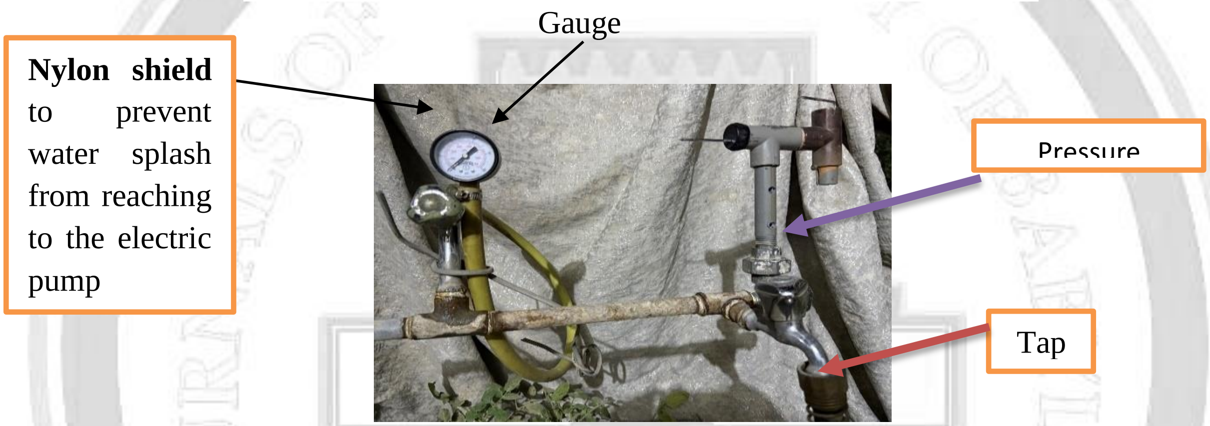


Figure 39 real test experiment for pressure switch

Results of the test:

According to the way of test above, eight pressure values is selected to measure their corresponding displacement values. The test is repeated for three time and the displacement data was obtained in each time. Then, an average of the displacement is computed. The data obtained from the test is shown in Table 1 below

Table 1 pressure values and their corresponding average displacements

Gauge pressure KPa	Average Displacement mm
0	0
50	0
100	2.3
150	3.4
200	5.5
250	6.6
300	7.5
350	8.6

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلـة جـــامعة بـــابـل للعلـــوم الهندسية
Vol. 33, No.4. \ 2025
ISSN: 2616 - 9916

The chart shown in Figure 40 below shows the relation between the pressure and the displacement according to Table 1 above.

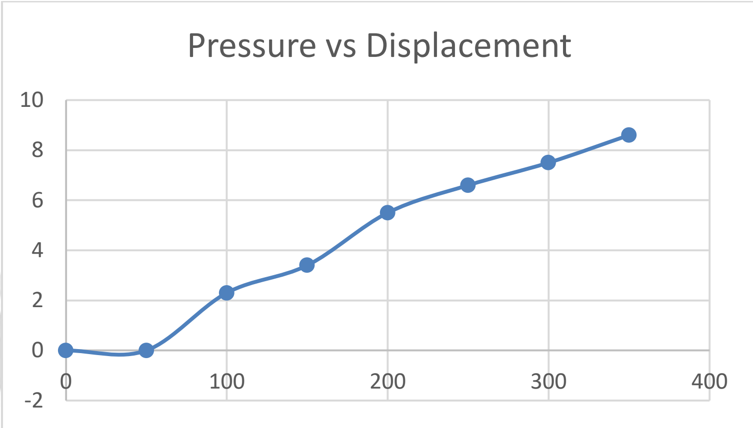


Figure 40 Pressure KPa vs Displacement mm

Results and Discussions of modeling and experimental results

The results obtained from the modeling and the practical test are shown in Table 2 below. Also, the chart shown in Figure 41 below shows the modeling and practical results together with applied pressure.

Gauge pressure KPa	Experimental Displacement mm	Modeling Displacement mm
0	0	-8.9373E-12
50	0	2.4647E-11
100	2.3	0.27021
150	3.4	2.1008
200	5.5	4.3346
250	6.6	6.774
300	7.5	8.3852
350	8.6	10.369

Table 2 pressure, experimental displacement, and modeling displacement

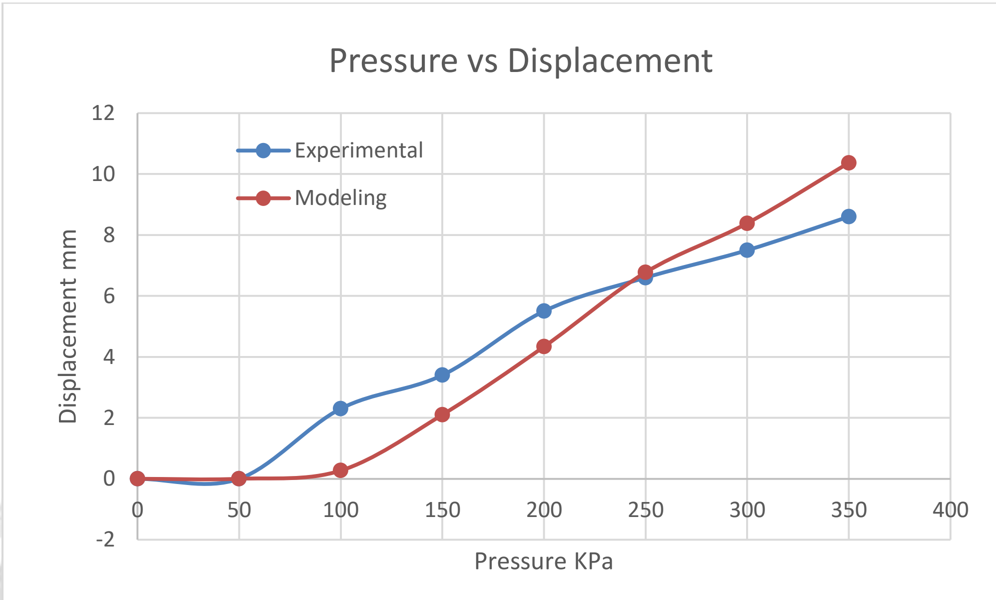


Figure 41 relation between pressure and displacement in modeling and in experiment

From the chart shown in Figure 41 above, there is noticeable difference between modeling and experimental results. In modeling with workbench 2022, the displacement has negative values at the beginning where very low pressure or zero value pressure is applied. That is because there is pretension in strip resulting in stretch, which previously applied in the strip and this pretension will move the leverage a little down.

Both the experimental and modeling or theoretical displacements generally do not influence pressure at the beginning because the applied pressure values at the beginning are low and cannot overcome the pretension resulting in stretch in strip. Then, by increasing the applied pressure, the pretension is overcome by high applied pressure and then, spindle together with leverage start to move up.

It can be seen the experimental displacement firstly or earlier starts to increase than in modeled displacement and that is because, in the modeling with Workbench 2022, the material property used in modeling is ideal while in experiment, the material is rubber parts formed and cut from an old tire's tube of truck and it is known that the mechanical specifications of this real rubber are degraded with time and environmental influences [18].

Also, for the same reason above, the displacement readings in the experiment are higher at pressure values of a range 70-200 KPa. But when the pressure reaches 250 KPa, the experimental readings start to be lower and continue to lower more with pressure increments. This case because friction among switch's parts becomes higher as pressure increases and the parts (leverage and spindle) become no longer orthogonal which lead to form higher reactions and normal forces on frame and between those two parts themselves. These increments in normal forces and reactions increase the friction forces. While in modeling in Workbench 2022, the friction forces are ignored with frame and between the leverage and spindle. the only considered friction force in modeling is between the diaphragm and spindle.

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

In experiments, there is another notice, which cannot be referred in charts or tables, which is the displacement cannot instantaneously respond to the change in pressure but it takes several seconds i.e. when the pressure increases or decreases, the corresponding change in displacement of leverage cannot be noted at once but it takes several seconds to move upward and downward. This case is because of friction forces which restrict the motion as well as the inertia influence delays the response or motion. The inertia influence is accompanied to all internal parts like leverage, spindle, diaphragm and strip.

About this improvised switch:

This improvised item is super simple and super cheap as well. It does not require heavy machine, like lathe machine, to be manufactured and its parts are available everywhere. This switch is safe and well waterproofed, there are many modifications made on this item to prevent the water from reaching upward where the electric contact is. The leak of water resulting in any diaphragm failure can be easily seen through the holes previously drilled in long pipe of the frame and through the second T shape fitting as was explained before. This switch can work with wide ranges of pressure and can be easily set. Also, it does require high professional technician or craftsman to build this item. So, this tool is practical for daily life use in all life's conditions especially in disasters like natural disasters, war disaster…etc. The only issue with this item is that the rubber parts, especially strip, loose its mechanical specification with time, so they need to be replaced. This might not be an issue when new rubber parts are used or specifically designed instead of those from an old tube but this will increase the cost and it will be against the targets of improvising the item. so, this paper works like a guide for primitive technicians and workers in workshops to manufacturing and producing this item

Any type of electric contact can be mounted on the switch top and attached to the second bent of leverage as shown in Figure 42 below, where limit switch is used as an electric contact.

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)
مـــجلــة جـــامعة بـــــابـل للعلـــــوم الهندسية

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

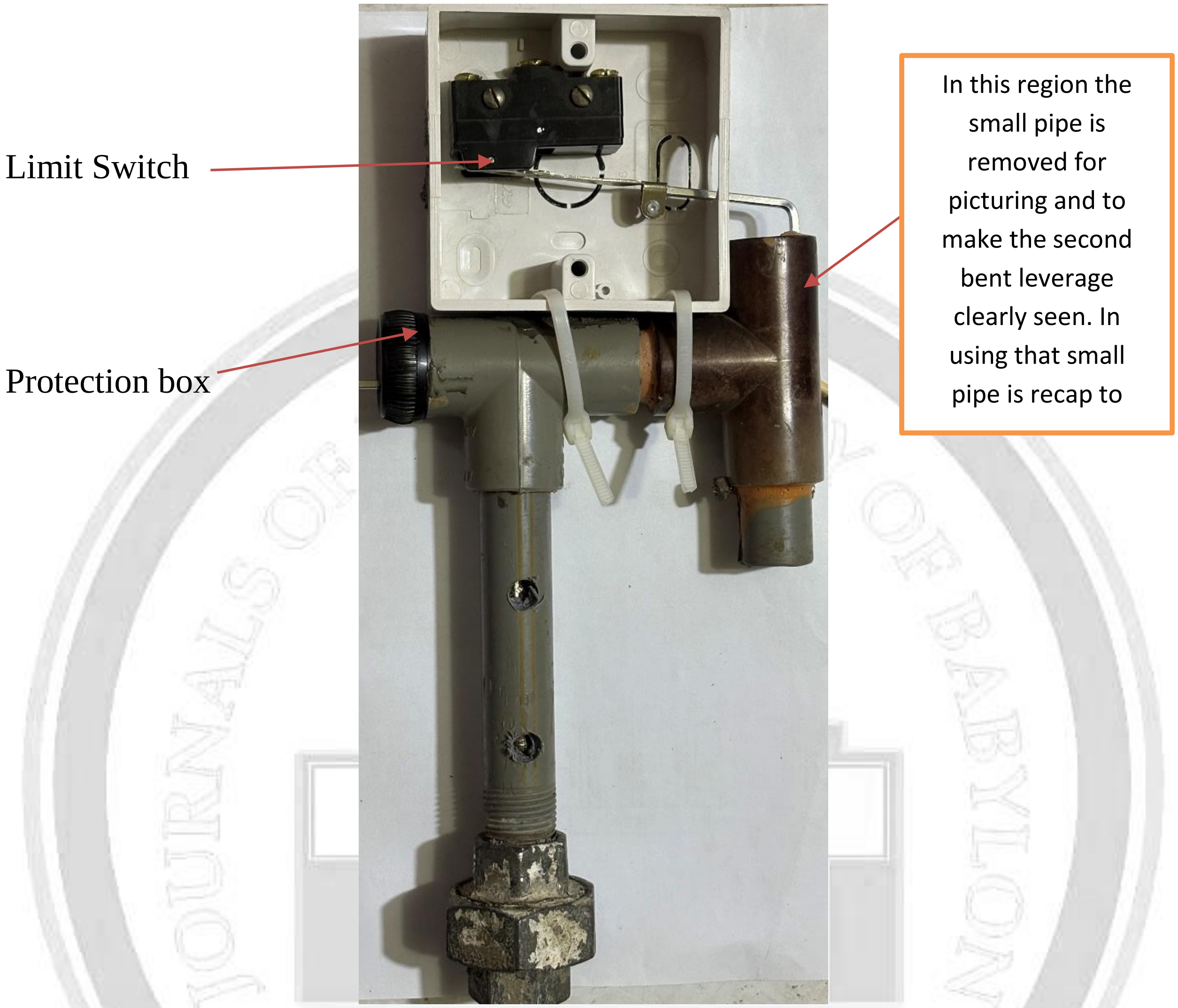


Figure 42 pressure switch with electric contact

Future work:

It will be to build special electric contact for this switch and to modify the switch to less number of moving parts to minimize the inertia and friction forces.

Reference:

[1] Tameson, “Pressure Switch”, [accessed: March 2025]

[2] TOM, “Pressure Switch”, Danfoss, [accessed: March 2025]

[3] TPC automation, “Pneumatic switch types and their applications” accessed: January 2025]

[4] Danfess “Pressure switches for air and water” Data sheet, Danfess, IC.PD.P10.H5.02 | 520B8125, July, 2017

[5] Hidra Matic “Technical Explanations for Mechanical Pressure Switches” Fuco, [accessed: March 2025]

[6] trp Polymer Solution company “rubber diaphragm” [accessed: April 2025].

[7] B.Mallikarjun, P.Dinesh, and K.I. Parashivamurthy “Finite Element Analysis of Elastic Stresses around Holes in Plate Subjected to Uniform Tensile Loading” *Bonfring International Journal of Industrial and Mangement Science*, Vol. 2, No.4, December 2012

[8] X.Chen and Y.Liu, Finite Element Modeling and Simulation with Ansys Workbench, USA, Taylor & Francis Group 2014

[9] Eskil Elgström”Practical implementation of hyperelastic material methods in FEA models”, M.S. thesis, Blekinge Institute of Technology, Karlskrona, Sweden, 2014

[10] H.Yang, R.Xiao, Z.Yang, D.Lei "Experimental characteristic and finite element modeling the deformation behavior of rubbers with geometry defects” *Science Direct* , Vol.80, December 2019

[11] L. R. G. TRELOAR “the physics of rubber elasticity” Great Britain, Oxford Classics Series, 2005

[12] Ahmed Al-barban “ Finite Element Analysis of a tube’s waste Rubber to be used as a Diaphragm in Pressure Switch that is being built from available cheap Materials” journal of the university of Babylon vol.31, no.5, pp.55-76, Aug.2023

[13] Ansys Workbench , material library, hyperelastic material specification, 2022

[14] MSC Software Corporation “Nonlinear Finite Element Analysis of Elastomers” MSC.Software: Whitepaper 2010

[15] Within Epsilon” Non Linear Adaptivity” Twin Cities Ansys User Meeting, June, 2018

[16] Alexander Hu & Brue Peachey” Redesigning an Experiment to Determine the Coefficient of Friction” Journal of Emerging Investigator, NewYork, June, 2016

[17] P.Arrozola, T. Ozal “Investigation on the effects of friction modeling in finite element simulation of machining”, *International Journal of Mechanical science*, Vol. 52, January 2010

[18] Ronald J. Schaefer, “MECHANICAL PROPERTIES OF RUBBER” NewYork, 2010

ARTICLE

JOURNAL`S UNIVERSITY OF BABYLON FOR ENGINEERING SCIENCES (JUBES)

مـــجلــة جـــامعة بـــــابـل للعلــــوم الهندسية

Vol. 33, No.4. \ 2025 ISSN: 2616 - 9916

التصميم والتصنيع الكامل لمفتاح الضغط الكهربائي من مواد رخيصة ومتوفرة بواسطة التحليل الشرائحي لبعض اجزاءه الغير معدنيه التي تعمل بشكل متزامن داخل مفتاح الضغط

احمد حمد يحيى البربن

قسم السياميك ومواد البناء، كلية هندسه المواد، جامعة بابل، **بابل, العراق**

Mat.ahmed.hamad@uobabylon.edu.iq

الخلاصه:

تم استخدام خرده بلاستكيه و مطاطيه في صناعة مفتاح ضغط بالإضافة الى القطع المعدنيه. ان تصميم مفتاح الضغط هذا راعى استخدام ارخص المواد وكذلك راعى البساطه بحيث أي تقني مبتدئ يستطيع صناعة هذه الالة. علاوة على ذلك, تصميم هذا المفتاح راعى أيضا المخاطر الناجمه من فشل الفاصل المطاطي (الدايفرام). ابعاد قطع وأجزاء هذا الجهاز مبنيه على حسابات باستخدام التحليل الشرائحي المحدد لقطعتي المطاط اللتان تعملان بالتزامن. القطعه المطاطيه الأولى تعمل كفاصل مطاطي (دايفرام) والثانيه التي هي عباره عن شريحه تعمل كنابض وكاداة تنظيم الشد اعتمادا على قيمة معينة للضغط. أيضا هنالك قطعتين معدنيتين بمفتاح الضغط لنقل الحركة من الفاصل المطاطي(الدايفرام) الى الشريحه (القطعه المطاطيه الثانيه). القطعه المعدنيه الأولى هي الساق الميكانيكيه والثانيه هي العتله المربوطه بالشريحه المطاطيه وأيضا مربوطه بنقاط التلامس الكهربائيه. تم رسم القطعتين المطاطيتين إضافة الى القطعتين المعدنيتين ببرنامج الكاتيا Catia R17 وبعدها تمت تحميل جميع القطع المرسومه على برنامج الانسس Ansys Workbench 2022 حيث تم عمل نموذج رياضي ومحاكاة لمعرفة العلاقه بين الضغط المسلط على الفاصل المطاطي (الدايفرام) و الاستطاله في الشريحه المطاطيه. واعتمادا على النتائج المستحصله من المحاكاة تم بناء مفتاح ضغط حقيقي وتم اختبار هذا المفتاح بتسليط نفس الضغوط المسلطه بالمحاكاة للحصول على استطالات عملية ومقارنتها مع نظيراتها المحسوبه نظريا بالمحاكاة. حيث تبين ان الاستطالات العملية تكون الأعلى عند الضغوط الواطئه لان الخواص الميكانيكيه للمطاط الحقيقي القديم تقل مع الزمن بينما بالضغوط العاليه , الاستطالات العمليه تكون اقل بالمقارنة مع نظيراتها بالمحاكاة لان الاحتكاك يكون عالي بالمفاصل عند الضغوط العاليه خصوصا انه تم اهمال الاحتكاك بالمفاصل اثناء المحاكاة. عموما رغم هنالك اختلاف قليل بالنتائج العملية والنظريه لكنها منطقيه وتتبع مسار واحد

الكلمات الدالة: اجهاد، انفعال، معايرة منحني المطاط ، تقطيع شرائيحي تلقائي اثناء الحل، تشوهات، احتكاك، تسليط ضغط ، تقارب المحني للوصول للحل، عمل مفاصل ببرنامج الانسس.