Study of the Mechanical Behavior of an Automotive Clutch Pedal Produced by Injection Molding

José Augusto da Estrela Inácio

jose.aeinacio@gmail.com

Instituto Superior Técnico, Universidade de Lisboa, Portugal

June 2019

Abstract

The mechanical behavior of a fiber-reinforced polymer material part cannot be correctly analyzed simply through a structural simulation without considering its mechanical properties and the geometric changes resulted from the manufacturing process, which can be determined through injection process numerical programs. These programs not only provide predictions about the fiber orientation of the polymeric parts, but also determine the residual stresses that coexist in them after the manufacturing process, the residual extensions and the location of welding surfaces. For a correct structural simulation, it is necessary to include in this analysis all the variables resulting from the manufacturing process, using an interface that correctly performs the mapping of the variables in the data of the structural analysis.

In this work it is studied the mechanical behavior of a clutch pedal produced by injection molding in the company Toolpresse, Ltd. The computational simulation of the injection process was performed through the Moldflow program, while the mechanical response of the pedal to experimental tests in a bench test was analyzed by the Abaqus numerical simulation program. The data transfer from the injection molding simulation, e.g., values of deformations, residual stresses and fiber orientation, to the structural mesh was performed through the Helius PFA interface program. The quality of the data mapping from the injection molding simulation to the structural mesh is analyzed in this work through comparisons between the numerical and experimental results, regarding the force required to deform the pedal in the test bench.

Keywords: Injection Molding, Moldflow, Abaqus, Helius PFA, Mapping, Fiber Orientation

1. Introduction

Polymeric materials have increasingly been replacing the metals used in components in the automotive industry, in particular fiber-reinforced polymeric materials. However, these materials have an anisotropic behavior, which means they have different mechanical properties depending on the loading applied. It is therefore important to correctly predict the mechanical behavior of the material for certain requests, through numerical simulation software such as Ansys or Abaqus, giving the polymer material characteristics that come from injection software such as Moldflow or Moldex3D.

The data transfer from the polymeric material to the structural simulation can be done through appropriate software such as Helius PFA or Digimat. Since these interfaces are still recent, it is not known, for geometrically complex parts, if the data transfer will be successful, once there can be errors in the mapping of injection results.

The softwares used in this work were: Autodesk Moldflow Insight 2019, for the injection molding process simulation; Simulia Abaqus 2017, for the structural analysis simulation; and Autodesk Helius PFA 2019 is the interface for the data transfer of the injection molding process simulation to the structural mesh.

2. Injection Molding

Injection molding is a production technique that manufactures products from polymeric materials.

2.1 – The Injection Cycle

It is a cyclic process involving the following steps [1]:

• Injection - The molten polymeric material is then injected to the mold cavity due to the movement of a screw.

• Compression - Upon reaching 95 to 99% of the cavity fill, the pressure is controlled in order to add more material to compensate the shrinkage of the part resulting from its continuous cooling. This phase is done after the gate is completely solidified.

• Cooling - The mold is provided with channels in which a coolant liquid flows, in order to accelerate the cooling of the molded part.

• Ejection - after the part being completely solidified, the mold is opened and the part can finally be ejected by the action of ejector pins.

Parameters associated with these steps of the injection molding process need to be strictly defined by the manufacturer in order to achieve an optimum cycle time, with reduced warpage and keeping the mechanical properties of the part.

2.2 – Residual Stresses

The residual stresses are internal stresses retained in the material resulting from the injection molding process and are responsible for the warpage that occurs after the part being manufactured. The residual stresses resulting from injection molding process may appear due to thermal variations in the part thickness or due to phenomena related to the polymeric material flow [2].

3. Case Study

The component studied in this work is an injection molded produced automotive clutch pedal for a model of the Renault brand. The material used is Technyl® A 218 V35 Black 21N, polyamide of the family PA66, thermoplastic, with 35% glass fiber reinforcement (GF35). Figure 1 shows the CAD representation of the clutch pedal for two opposite views.

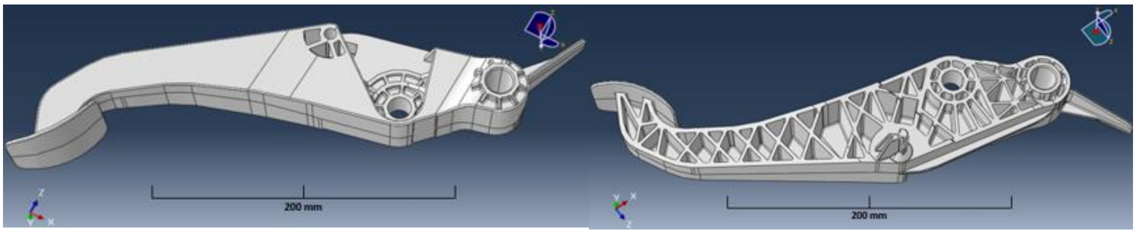

Figure 1 – CAD geometry of the clutch pedal

This pedal is overmolded on a metallic component that attaches the pedal to the rod of the hydraulic circuit cylinder. During the filling phase, the molten polymeric material will surround this metallic insert, and after solidification both components will be coupled, behaving as a single part. Figure 2 illustrates the clutch pedal attached to the metallic insert.

Figure 2 – Metallic Insert

It is possible to note, through the two previous figures, that the CAD geometry in the hole of the clutch pedal that overlaps the metallic insert has been modified to a single cylindrical bore, comparing with the same hole of the actual pedal that follows the geometry of the metallic insert surface.

3.1 – Applied Forces

In order to move the clutch pedal, the driver needs to apply a force on the pedal foot. This force depends on the pedal displacement, but generally the maximum force that the driver needs to apply occurs in the maximum displacement position, and does not reach 200 N (or 20 kgf) [3].

The pedal displacement is limited by the hydraulic circuit cylinders stroke, i.e., the stroke end of the cylinders dictates the maximum pedal displacement. Until the pedal reaches the maximum displacement position, increasing the applied force will promote pedal movement. If this position is reached, and if the driver does not decrease the force applied, the pedal will not retreat and it will tend to deform due to a bending moment caused by the reaction force of the hydraulic circuit cylinders that opposes the actuating force of the driver, according to Figure 3.

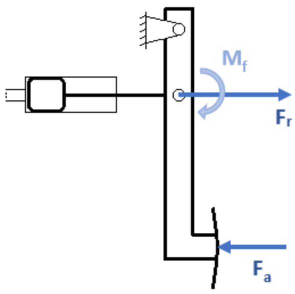

Figure 3 – Pedal forces and moments in maximum displacement condition

The clutch pedals need to be manufactured in such a way that can resist these stresses cyclically.

3.1 – Experimental Tests

In order to test the mechanical strength of the clutch pedal, loading tests were performed under similar conditions in which the pedal is subjected when in maximum displacement.

The experimental tests were performed in the Toolpresse, Ltd. company, that disposes a testing bench in which the pedal is constrained and deformed by the linear movement of a cylinder, which simulates the force applied by the driver, with a certain scale factor, that will bend the pedal. It is ensured that the pedal is fixed by screwing it through the pivot hole (pedal center of rotation) to the testing bench and supporting the spherical surface of the metallic insert.

Three different types of tests were performed. The first one was a loading test until the pedal reached the rupture, in order to know the maximum force required, as well as to locate the beginning of the crack and its propagation along the pedal. In this test, the cylinder moved approximately 40 mm from the start of the contact, and it was detected a maximum force of 256 kgf before the rupture. The fracture occurred at the traction side near the metallic insert, and propagated to the opposite end, as shown in Figure 4. Therefore, it can be considered, from now on, that the critical region of the clutch pedal is near the metallic insert.

Figure 4 – Clutch pedal rupture

The second test was a loading test near the rupture limit, with immediate recoil, to get an idea of the maximum plastic deformation that the pedal can withstand before breaking. In this test, the cylinder traveled approximately 27 mm and the load cell detected 240 kgf before the start of the recoil. The plastic deformation due to this loading was approximately 5 mm.

The third test consisted of multiple and consecutive loading and unloading (eleven each). At the first load the cylinder obtained a maximum force of 40 kgf, while the last one obtained a maximum force of 240 kgf. The total cylinder displacement was approximately 25 mm instead of 27 mm from the previous test, and the total plastic strain was approximately 4 mm instead of 5 mm.

4. Injection Molding Process Simulation

For the simulation of the injection molding process it was applied onto the clutch pedal the material Technyl A 218HPX V35 Black21N, being the most similar material, comparing to the one used in the actual manufacture – Technyl A 218 V35 Black21N. For this material, Moldflow recommends values for some of the injection parameters, according to Figure 5, being necessary to highlight the surface temperature of the mold and the ejection temperature.

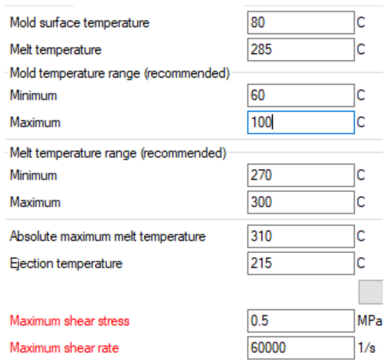

Figure 5 - Moldflow recommended values for Technyl A 218HPX V35 Black21N

Initially, Moldflow assigns the material to both parts (pedal and insert), since it assumes that all CAD geometries are parts that will be injected. Therefore, Moldflow must recognize the metallic component as a part insert, and attribute its mechanical properties.

It was needed to generate the mesh of the clutch pedal and its metallic insert, such as it was necessary to create the geometries of runners and cooling channels according to the mold CAD provided by the Toolpresse, Ltd. company, as Figure 6 shows.

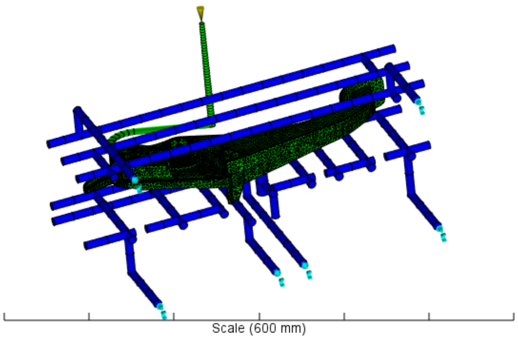

Figure 6 – Runners and cooling circuits

The real mold injects two pedals at the same time, but it was simulated the injection molding process of only one, in order to simplify the analysis.

4.1 – Process Settings

According to other data provided by the Toolpresse, Ltd. company, it was possible to calculate the variation of flow rate of the molten polymeric material with the position of the injection machine screw in the numeric simulation, knowing that the flow rate and the volume injected in the simulation needs to be about half the values of the real process, once there is only one pedal on the simulated process, instead of two. Figure 7 shows the profile of the simulated screw position vs flow rate.

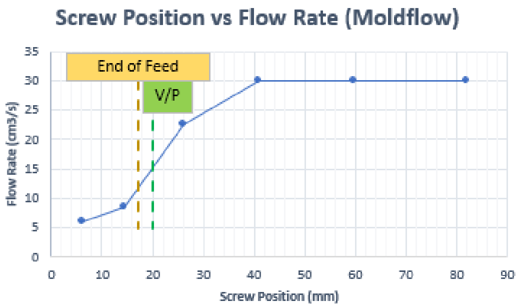

Figure 7 –Screw Position vs Flow Rate in the simulation of the injection molding process

In the filling analysis, there was a difference of about 6% to the switching instant of the process control from speed to pressure (V/P instant) obtained (89.6%), comparing with the expected value (95.7%).

In order to diminish this difference, it was decided to calculate a new initial position of the screw, so that in the simulation of the injection molding process, the V/P instant occurs at 95.7%. To compensate for excessive screw feed, its starting position was moved backwards, so that the volume to be injected was virtually higher than the injection volume previously calculated.

The results of the deformations after the simulation of the injection molding process will be compared with the deformations of the structural simulation and discussed in chapter 7.

5. The Structural Model

The file imported into Abaqus was a different STEP file than the one used in Moldflow. It was used a STEP file with the clutch pedal CAD on the testing bench of the company Toolpresse, Ltd. so that the simulated bending tests have a good geometric approximation to the experimental ones, knowing the position and the movement of the actuator cylinder. The testing bench was only necessary to create the geometry of the actuator cylinder, and to place it correctly in the assembly. Once this positioning was done, the testing bench parts were deleted from the assembly.

Among the experimental tests performed, the tests that were simulated in Abaqus were the ones that did not lead to the fracture of the pedal, i.e., both tests with a maximum force of 240 kgf - single load; and multiple loads with increased force. However, in the multiple loading simulation, the pedal was only loaded four times, instead of the eleven verified in the experimental test, with the cylinder displaced the equivalent of 40, 80, 120 and 160 kgf. The loads were controlled so that the displacement of the cylinder of the structural simulations was similar to the displacement of the experimental tests.

5.1 – Steps

For each step it is necessary to indicate its duration, the maximum number of increments, the value of the first increment of time and also the maximum and minimum possible value of the remaining increments.

The first step to be created is the step 0, which is the stage in which the final deformations resulting from the injection molding process of the pedal, until the residual stresses are stabilized, thus obtaining the geometry of the pedal after its manufacture. At this stage, the pedal must be deformed freely, i.e., it cannot be carried out with the application of boundary conditions on the clutch pedal. In this step the "Nlgeom" option is also activated, so that the simulation considers the geometric non-linearity, which must be implied when permanent deformations are involved. Abaqus provides a stabilizing mechanism for quasi-static models thanks to a damping factor, that in this study was defined as a dissipated energy fraction.

The remaining steps refer to the simulation steps of the pedal bending tests, so these will be different by the type of test performed. For the bending test in which it was intended to load and unload the pedal once, only two steps were imposed, one step for loading and another for the unloading. For the incremental test, eight steps were required, one step for each loading, and another step for each unloading. The end of each loading step will be dictated when the maximum displacement of the cylinder reaches approximately the same value as the maximum displacement of the equivalent experimental load.

It is considered the beginning of the cylinder displacement when the cylinder contacts the pedal first, therefore, it is necessary to add to the step time the lead time until the beginning of that contact.

5.2 – Boundary Conditions

In order to apply the boundary conditions on the pedal, it is important to correctly interpret the pedal constraints on the testing bench. It was considered, therefore, that in the structural simulation in the hole of the clutch pedal is encastred, and the spherical head of the cylinder is thus considered to be a fixed support of the clutch pedal.

Boundary conditions have also been applied to the actuator cylinder since it can only move in the normal direction to its contact face with the pedal. Therefore, the cylinder cannot move parallel to this face, nor can rotate in any direction. Therefore, a coordinate system was created aligned according to the directions of the cylinder, and the fastening conditions and speeds imposed were applied according to these directions. The chosen speed was the same for all the loading and unloading, with a value of 0.087 mm/s, which was the velocity value at the 160 kgf loading of the experimental incremental test. This speed takes opposite directions to the loading and unloading steps.

The results for the step 0 and the various loadings performed will be exposed and compared in Chapter 7.

6. Advanced Material Exchange

Helius PFA is a software that allows to map results of injection molding simulation, performed in Moldflow, to a mesh of a structural analysis, through a tool named Advanced Material Exchange (AME). This mapping will make the structural analysis more complete, since this will consider the influence of the fiber orientation, as well as the total deformations after the ejection of the manufactured part. It also allows the import of the tension and the compression characteristic curves of the composite material, for various fiber orientations, being able to later export them to the structural software, as well as to the injection molding software.

For this study it was not possible to carry out the tensile tests to obtain the proper material characteristics for the different orientations required (0º, 45º and 90º), since the test samples didn't arrive in time. As such, the solution was to look for a tensile test of a similar material. Figure 8 shows the curves of a PA66-GF35 (35% glass fiber) with 0% relative humidity (DAM), for the three required orientations, whose trade name was not specified.

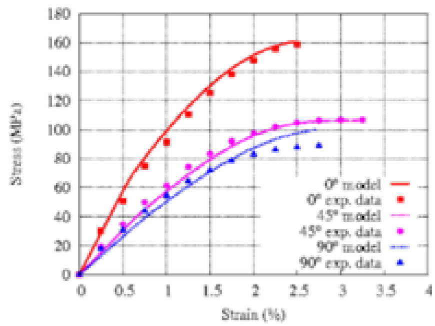

Figure 8 – Tensile curves of a PA66-GF35 material for three fiber orientations [4]

6.1 - Theoretical Fundamentals

6.1.1 – Simplifying Assumptions

The Advanced Material Exchange has developed a multiscale material model based on the following assumptions and constraints [5]:

• The fibers only present a simple linear elastic response;

• The matrix constituent material undergoes plasticity and fracture;

• The plasticity and fracture of the matrix of the model in question must consider the detachment of the fiber with the matrix that occurs in the material;

• The nonlinearity of the matrix material is responsible for the non-linearity exhibited by the composite material;

• The plasticity and fracture of the matrix constituent are driven by their own stresses, rather than being driven by the homogenized stresses of the composite material;

• The nonlinear responses of the matrix constituent are strongly dependent on the degree of the fiber alignment;

• As the degree of fiber alignment increases, the plasticity and fracture responses of the matrix constituent become strongly dependent on the loading direction relative to the average fiber direction.

6.1.2 – Material Homogenization: The Eshelby-Mori-Tanaka Criterion

This criterion is based on the Eshelby criteria, where an inclusion is contained in a matrix, and both bodies are subject to the same strain field ε_0 [6]. The inclusion stress is thus obtained,

$$\sigma_f = C_f[\varepsilon_0 + \bar{\varepsilon} + \varepsilon(x)] = \\ = C_m[\varepsilon_0 + \bar{\varepsilon} + \varepsilon(x) - \varepsilon^*] \quad (1)$$

where $\bar{\varepsilon}$, $\varepsilon(x)$, and ε^* are respectively, the matrix, inclusion and applied strains, and C_m is the stiffness tensor of the matrix. The stiffness tensor for composite materials is,

$$C = C_f + f\left(C_f - C_m\right)A \quad (2)$$

where C_f is the stiffness tensor of the fiber, f is the fiber volume fraction and A the strain concentration tensor,

$$A = T[(1-f)I + fT]^{-1} \quad (3)$$

where,

$$T = \left[S_{eq}\left(C_{eq}\right)^{-1}\left(C_f - C_{eq}\right) + I\right]^{-1} \quad (4)$$

is the is the fourth order tensor that relates the average strain of the fiber to the mean strain in the matrix, and C_{eq} e S_{eq} are, respectively, the stiffness and Eshelby tensors.

7. Results

7.1 - Step 0

The deformation after the step 0 in Moldflow and Abaqus show a good correlation of values, where the largest difference in deformation values between the two software, about 1 mm, lies at the opposite end of the clutch pedal foot. However, this is the least critical region of the pedal since this is the only region that will be free of external forces.

Figure 9 shows the residual stresses obtained in Abaqus after the step 0 simulation. These results indicate that there are some regions near the metallic insert location, where these are higher than the data of the mechanical behavior curves of the material (see Figure 8). This leads us to consider that the mapping of the residual stresses was not perfectly calculated by the AME.

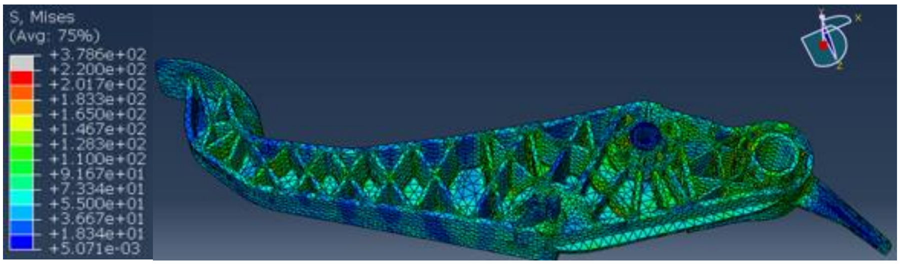

Figure 9 – Clutch pedal von Mises residual stresses after step 0

7.2 – Bending Test Simulations

Figure 10 illustrates the numerical and experimental evolution of the force with its displacement for the singular loading test and its unloading.

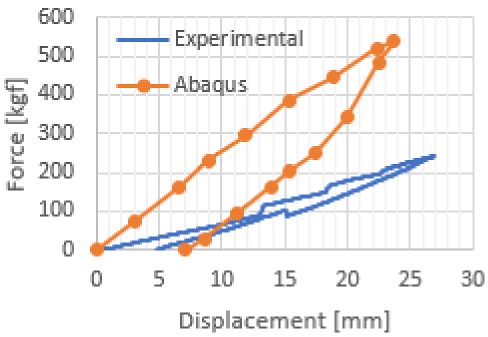

Figure 10 - Force vs displacement of singular loading tests

The major difference between the experimental test and the numerical results are the values of the actuation force of the cylinder. The simulated cylinder applies more than double the force than the experimental test cylinder along both profiles. Although this difference exists, there is not a great disparity between the plastic deformation values of the composite material after the end of the test, since in the experimental test the pedal deformed 5 mm in the direction of the cylinder movement, whereas in the simulation the deformation was 7 mm.

Figure 11 illustrates the numerical and experimental evolution of the force with its displacement for the multiple loading tests.

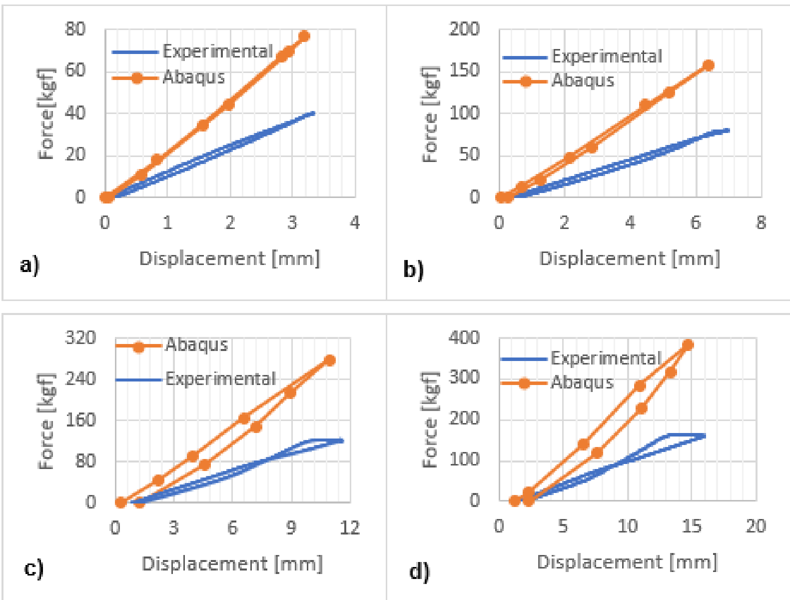

Figure 11 - Force vs Displacement of multiple loading tests; a) 40 kgf; b)80 kgf; c) 120 kgf; d) 160 kgf

As in the single-load test, it is noted that the force required to deform the cylinder in the multiple load numerical simulation is more than twice the strength of the experimental test for the four loads compared. The plastic deformation at the end of each loading was also well replicated in the simulation, with the largest difference being noted in the load equivalent to 80 kgf (about 0.4 mm between the simulation and the experimental test).

8. Conclusion

When transferring data from the manufacturing analysis to the structural analysis, it was found that, although the final deformations were similar before and after the mapping, the analysis of the effective stress values after step 0 allowed to identify values higher than the data provided relative to the mechanical behavior curves of the material, in the area of the metallic insert bore. These results, associated to the difference in the force values required for the pedal deformation obtained between the experimental tests and their numerical simulations, suggest that the mapping performed by the Helius PFA interface, which may not be unaware of the geometrical complexity of the pedal, could be the main reason for the difficulties encountered in the attempt of replicating computationally the experimental tests.

9. References

[1] I. Pires, “Moldação por Injecção: parte 1,” [Online]. Available: https://fenix.tecnico.ulisboa.pt/downloadFile/845043405457573/injeccao1.pdf.

[2] Santa Clara University, “Santa Clara University Engineering Design Center,” [Online]. Available: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/physics/b3400001.htm.

[3] E. Olmeda, S. F. Del Toro, G. María, J. P. Redondo and V. Díaz, “Clutch Pedal Sensorization and Evaluation of the Main Parameters Related to Driver Posture”, August 24, 2018.

[4] A. Launaya, M. H. Maitournama, Y. Marcob e I. Raoultc, “Multiaxial fatigue models for short glass fiber reinforced polyamide. Part II: Fatigue life estimation.,” August 22, 2012

[5] Autodesk, Inc., “Advanced Material Exchange Help: Theory Manual,” [Online]. Available: http://help.autodesk.com/view/ACMPAN/2019/ENU/?guid=GUID-D60A05C0-DD63-47B6-B1CD-248F2BDF1A9D.

[6] B. N. Nguyen e J. Paquette, EMTA’s Evaluation of the Elastic Properties for Fiber Polymer Composites Potentially Used in Hydropower Systems, 2010.