



# CHALMERS

---



## Modelling of a Lock System in a Door Subject to Break-in with Crowbar

Master's thesis in Applied Mechanics

BRITTA KÄLLMAN



MASTER'S THESIS IN APPLIED MECHANICS

# Modelling of a Lock System in a Door Subject to Break-in with Crowbar

BRITTA KÄLLMAN

Department of Applied Mechanics

Division of Dynamics

CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2017

Modelling of a Lock System in a Door Subject to Break-in with Crowbar  
BRITTA KÄLLMAN

© BRITTA KÄLLMAN, 2017

Examiner: Mikael Enelund  
Supervisor: David Fredriksson, CAE Value

Master's thesis 2017:26  
ISSN 1652-8557  
Department of Applied Mechanics  
Division of Dynamics  
Chalmers University of Technology  
SE-412 96 Göteborg  
Sweden  
Telephone: +46 (0)31-772 1000

Cover:  
Picture from physical break-in test at ASSA ABLOY.

Chalmers Reproservice  
Göteborg, Sweden 2017

Modelling of a Lock System in a Door Subject to Break-in with Crowbar  
Master's thesis in Applied Mechanics  
BRITTA KÄLLMAN  
Department of Applied Mechanics  
Division of Dynamics  
Chalmers University of Technology

## ABSTRACT

ASSA ABLOY [1] is today the global leader in door opening solutions dedicated to satisfying end-user needs for security, safety and convenience. Their lock systems are certified together with different doors against break-ins according to SS-EN 1630:2011+A1:2016 [2]. This standard specifies how long time the tests maximum should take and what tools that are allowed to be used. These tests are, however, very expensive though both a new door and a new lock system are destroyed. The purpose of this thesis is therefore to develop a method for analyzing the lock system in a door with the help of commercial computational software in the specific case of a break-in with a crowbar.

A model of one of ASSA ABLOY's lock system and a steel door with glass window are created in the commercial Multi-Body Dynamics software Adams. A convergence study is performed on the model with help of optimization methods in the software HEEDS. This model's purpose is to compute stresses and deformations in the lock system and from that analyze sensitive and exposed areas. With further development the model can be used to do a design optimization on the lock system.

To get a better visualization of the break-in simulation the MaxFlex tool in Adams, a tool that can handle non linearity, is introduced. The knowledge about the tool is very limited from the beginning and after some simulations a conclusions that Adams is not the optimal software for this problem is drawn. Without any movements and mechanisms in the model a commercial Finite Element software is more appropriate due to shorter simulation time. Adams is adapted to relatively short time steps which is needed when movements and mechanisms are involved, but unnecessary for this model. A new model of the lock system is created in the Finite Element software Abaqus. This model is compared to measured data from a physical test and it can be seen that the deformation on the lock system has similar shape as in the physical test, but with a smaller total deformation.

Keywords: Lock system, Door, Crowbar, Finite Element Analyze, Adams, Abaqus, Optimization, HEEDS



## PREFACE

This project has been performed the spring of 2017 as a Master Thesis at the program Applied Mechanics at Chalmers University of Technology. The project has been done for the consulting company CAE Value in cooperation with ASSA ABLOY. David Fredriksson, Joel Albrektsson and Kim Bladh at CAE Value have supervised the project. The examiner and supervisor from Chalmers Univeristy of Technology has been Mikael Enelund.

## ACKNOWLEDGEMENTS

Firstly, I would like to thank ASSA ABLOY for the project concept and for letting me be a part of your work for making the lock system safer and hence make homes safer for everybody. An extra thank to Mathias Fält and Fredrik Pärus at ASSA ABLOY for all their help and input to the work. Many thanks to David Fredriksson, Joel Albrektsson and Kim Bladh at CAE Value for all their help and patience during my thesis. At last a wish to thank my examiner at Chalmers University of Technology, Mikael Enelund, for great support during the whole project.





# NOMENCLATURE

## Abbreviations

BDF	Bulk Data File
CAE	Computed Aided Engineering
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Method
GUI	Graphical User Interface
MBD	Multi Body Dynamics
MBS	Multi Body Systems
MNF	Modal Neutral Files



# CONTENTS

<b>Abstract</b>	<b>i</b>
<b>Preface</b>	<b>iii</b>
<b>Acknowledgements</b>	<b>iii</b>
<b>Nomenclature</b>	<b>v</b>
<b>Contents</b>	<b>vii</b>
<b>1 Introduction</b>	<b>1</b>
1.1 Background . . . . .	1
1.2 Purpose . . . . .	1
1.3 Objectives . . . . .	1
1.4 Delimitations . . . . .	1
1.5 Company presentation . . . . .	1
1.6 Literature study . . . . .	2
<b>2 Theory</b>	<b>2</b>
2.1 Optimization . . . . .	2
2.1.1 SHERPA . . . . .	3
2.2 ASCII . . . . .	3
2.3 ANSA . . . . .	3
2.4 Adams . . . . .	3
2.4.1 Adams MaxFlex . . . . .	4
2.5 MSC Nastran . . . . .	4
2.6 Abaqus . . . . .	4
<b>3 Geometry</b>	<b>4</b>
3.1 Lock system . . . . .	4
3.2 Door and door frame . . . . .	8
<b>4 Loading case</b>	<b>9</b>
<b>5 Method</b>	<b>10</b>
5.1 Rigid Body model . . . . .	11
5.1.1 Geometry . . . . .	11
5.1.2 Connections, contacts and forces . . . . .	11
5.2 Mixed flexible and rigid body model . . . . .	13

5.2.1	Geometry . . . . .	13
5.2.2	Clean up and meshing of flexible bodies . . . . .	13
5.2.3	Adams model . . . . .	15
5.3	Convergence studies . . . . .	18
5.4	HEEDS structure . . . . .	19
5.5	MaxFlex model . . . . .	21
5.5.1	Striking plate . . . . .	21
5.5.2	Door frame . . . . .	22
5.6	Abaqus model . . . . .	23
5.7	Verification of Abaqus model of the lock system . . . . .	25
5.7.1	Measure data . . . . .	27
<b>6</b>	<b>Results</b>	<b>28</b>
6.1	Mixed flexible and rigid model . . . . .	28
6.2	Convergence studies . . . . .	28
6.2.1	Time step size . . . . .	28
6.2.2	Element size . . . . .	30
6.3	Non linear material model in Adams - MaxFlex . . . . .	31
6.3.1	MaxFlex model of striking plate . . . . .	31
6.3.2	MaxFlex model of door frame . . . . .	32
6.4	Verification of Abaqus model of the lock system . . . . .	33
<b>7</b>	<b>Discussion</b>	<b>35</b>
7.1	Software . . . . .	35
7.2	Verification of Abaqus model . . . . .	35
<b>8</b>	<b>Future work</b>	<b>36</b>
	<b>References</b>	<b>37</b>
<b>A</b>	<b>BDF for MNF generation of the flexible bodies</b>	<b>I</b>
<b>B</b>	<b>Simulation script used in HEEDS for running the Adams simulation</b>	<b>II</b>
<b>C</b>	<b>BDF parameters for the MaxFlex model of the striking plate</b>	<b>III</b>
<b>D</b>	<b>BDF parameters for the MaxFlex model of the door frame</b>	<b>IV</b>

# 1 Introduction

This project consist of modeling a lock system in a door subjected to break-in. In this chapter it is presented what the benefits of this kind of modeling could be, the objectives of the project and what limitations that are assumed.

## 1.1 Background

A break-in to ones home is today something that scares many people. A strengthening of the doors and lock system would therefore make people feel safer but it would also result in less police resources for solving break-in cases.

ASSA ABLOY [1] is today the global leader in door opening solutions dedicated to satisfying end-user needs for security, safety and convenience. Their lock systems are certified together with different doors against break-ins according to SS-EN 1630:2011+A1:2016 [2]. This standard specifies how long time the tests maximum should take and what tools that are allowed to be used. However, how the tools are to be used and with what force are not specified. This means that the test could be extremely different from time to time and it is difficult to predict the result.

Though ASSA ABLOY wants to make the locks even more safe for their customer and the fact that the tests are expensive and hard to predict the subject of this master thesis is to develop a novel method for analyzing and simulating lock systems with respect to break-in loads without the need of physical testing.

## 1.2 Purpose

The purpose of this master thesis is to develop a novel method to visualize, simulate and analyze the lock system in doors, in the specific case of break-in with a crowbar.

## 1.3 Objectives

The main goal is to obstruct the break-ins for thieves. By creating a method for visualizing, simulating and analyzing the lock system in a door in the specific case of a break-in with a crowbar, the lock system, together with the door, can be approved to handle the real physical test without actually doing one. This is one step on the way to improve the lock system since, with a complete model, the stress distribution and deflection can be calculated and exposed areas can be identified. To verify the model of the lock system the results from a simple loading case will be compared to measure data from a physical test done by ASSA ABLOY. To do all this the computer programs ANSA [3], Adams [4], MSC Nastran [5] and Abaqus [6] will be used and combined. Convergence studies will be done to verify the mesh size and time step size. The studies will be done in HEEDS [7] with the use of optimization methods.

## 1.4 Delimitations

The CAD-models for both the lock system and the door are input data but only the lock system is to be analyzed. The process only includes established computer programs but probably together with own written user script. The loading case is not specified exactly in the standard and needs to be limited and somehow defined. Multiple different tools are allowed according to the standard, but only the crowbar will be accounted for here.

## 1.5 Company presentation

CAE Value [8] is a consulting company specializing in optimization analytics and Multi Body Simulations (MBS). The company was founded in 2005 with the vision to accelerate the development and increasing the value of functional virtual prototypes.

## 1.6 Literature study

A pre-study to find earlier similar projects has been done. With searches for (“optimization” OR “calculation” OR “Finite Element”) AND “lock”, nothing similar was found. The only optimization projects found on lock system are focused on the keys and therefore not relevant for this project. Therefore the literature study is expanded further to similar applications and also background for methods to be used in this project.

In the standard for break-in in doors [2] it is not specified how and where the load should be applied, only what tools that are allowed and how long time they are allowed to take. If looking at standards on cars instead more detailed information can be gained. There are, for example, standards for *Side Impact Protection* [9] which is a standard for what maximum impact on the car that is allowed when it is hit with a rigid steel cylinder at the side doors. In this standard there are specifications on the impact of the door opening during and after a crash such as the door that is struck by the moving deformable barrier should not separate totally from the car. There are also specifications for any door not struck by the moving deformable barrier, these are that the door shall not disengage from the latched position, the latch shall not separate from the striker, and the hinge components shall not separate from each other or from their attachment to the vehicle and last, neither the latch nor the hinge systems of the door shall pull out of their anchorages. With more specific standards as this it is more straightforward to create finite element analysis that are conformable to the physical tests and therefore it is easier to find documentation on calculations on car doors than on house doors.

In [10] HEEDS is efficiently used to minimize the mass of a vehicle door with constraints on stress and displacement in five different loading cases. Three of these cases are in some way similar to this project though the force is applied on the doors latch. The first loading case is a vertical force on the latch when the door is fully open to control that the sag is not too large. The second loading case is a horizontal load applied to the latch, pushing the door into an over-open condition. The third loading case is the one most similar to this project, when a rotational acceleration is applied to the door with the latch constrained. The values of the constraints on the displacement and stress are not presented in this report but with given design variables (of geometry lengths) HEEDS generated a result of a mass reduction on 4.36 %.

HEEDS optimization process has been applied to several automotive lower compartment rail designs, where performance has been improved significantly and mass reductions of 20 % has been gained. This is compared to baseline rails designed by experienced engineers [11].

## 2 Theory

### 2.1 Optimization

Optimization is a method for finding the best possible solution [12] to, for example, a design problem with a given set of constraints. To do this it is important to formulate an effective goal (or *objective*), for example, what aspect of the design that need improvements. The objective is formulated from measurable values (individually or a combination of them) to be minimized or maximized. The optimization formulation must also include limitations or restrictions (or *constraints*) and could, for example, be limitations on the size of the product, how much force it must be able to handle or physical laws and relations.

For the verification process, based on an optimization method, the computer program HEEDS [7] will be used. HEEDS can be linked together with any commercial or proprietary CAE (Computed Aided Engineering) tool that creates input and/or output files in ASCII format (see Section 2.2). The default search method in HEEDS is SHERPA (see Section 2.1.1) which uses multiple search strategies simultaneously and also dynamically adapts to the problem as it learns about the design. HEEDS, in comparison to Matlabs optimization algorithms LSQNONLIN and FMINCON, provides a much user friendlier environment through its graphical user interface [13]. For the evaluation the setup in the program is the same as for an optimization process besides from that the study will be set to *Evaluation only* instead of *Optimization*. This means that for each evaluation the design variables are fixed and hence no optimization will be done. The study in HEEDS will connect multiple programs, a simplified flow chart for this is seen in Figure 2.1.

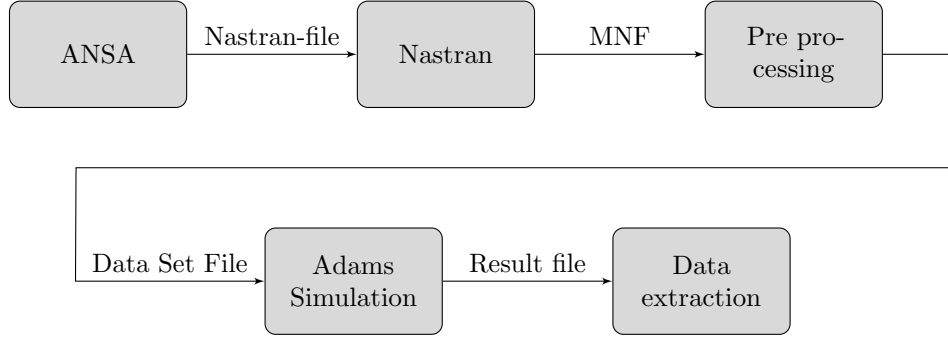


Figure 2.1: *Simplified expected flow chart of HEEDS structure.*

### 2.1.1 SHERPA

SHERPA stands for *Simultaneous Hybrid Exploration that is Robust, Progressive and Adaptive* [14]. SHERPA uses the elements of multiple search methods (both global and local methods) simultaneously in a unique mixture and therefore takes advantage of the best attributes of each method. Each participating approach contains internal tuning parameters which are modified automatically during the process according to knowledge gained about the design space. This knowledge also determines when and to what extent each approach contributes to the search. In other words, SHERPA adapts itself to do the search as efficient as possible, independent on how the design space is constructed.

The advantage of SHERPA is that it is fast (needs fewer iterations than most other algorithms), robust (identifies better-quality solutions) and no tuning parameters are needed, only the specific number of evaluations wanted to conduct [14]. In [15] five different optimization problems are solved with four different optimization methods: SHERPA, Adaptive Simulated Annealing (ASA), Genetic Algorithm (GA) and Sequential Quadratic Programming (NLPQL). The efficiency of the method can be measured in two ways, by the number of evaluations required to fully convergence and the quality of the solution at a given number of evaluations. In both those measures SHERPA was found to be at least twice as good as the next best method. Besides from this, all methods but SHERPA had a great variations in the performance depending on the problem. SHERPA however, is attributed to its hybrid and adaptive formulation which makes it robust over a wide range of problems.

## 2.2 ASCII

ASCII stands for *American Standard Code for Information Interchange* is a standard of character encoding [16]. It represents text in computers and other technical devices. More information on how ASCII arised, how it works and why it is designed as it is can be read in [17] and [18].

## 2.3 ANSA

ANSA is an advanced multidisciplinary CAE pre-processing tool which provides all the necessary functionality for full-model build up [3]. ANSA has everything from CAD data to ready-to-run solver input file, in a single integrated environment. It has an integrated Batch Meshing tool which leads to controllable and effortless optimal results, for both shell and volume meshing.

## 2.4 Adams

Adams is a Multibody Dynamics (MBD) Simulation software and therefore helps study the dynamics of moving parts, and how loads and forces are distributed throughout mechanical systems. The program offers early system-level design validation and complex interactions between disciplines can be evaluated and managed [4].

### 2.4.1 Adams MaxFlex

Adams MaxFlex is a tool in Adams which allows for the representation of geometric nonlinearity (large deformations), material nonlinearity and boundary condition nonlinearity. The tool's focus is to provide a solution for problem with both nonlinear behavior and motions and loads that influence each other and, therefore, accurate results are hard to gain through separate MBD and Finite Element Analyses (FEA) [19].

## 2.5 MSC Nastran

MSC (MacNeal-Schwendler Corporation) Nastran (hereafter referred to as Nastran) is an application used to perform static, dynamic, and thermal finite element analyses across the linear and nonlinear domains. It can also handle automated structural optimization and embedded fatigue analyses [5].

## 2.6 Abaqus

Abaqus is a FEA product suited for engineering problems in a great variation of applications, such as dynamic vibrations, MBS, crash simulations, nonlinear static and thermal coupling [6].

# 3 Geometry

In this chapter the origin model of the lock system and door are presented.

## 3.1 Lock system

The studied lock system is the electromechanical lock 850C-50 from ASSA ABLOY, see Figure 3.1. The lock is only examined in the locked state with the Hookbolt fully extended, and is, thus, not represented in any other state in this report. The lock system has approximately 35 different parts excluding screws. However, for the purposes of this project some parts are excluded since it is assumed that electric components and the drill protection plate do not affect the mechanical strength of the system. The model is then simplified further by merging parts to decrease the total number of parts. When merging parts together the friction between the parts is assumed to be of negligible importance and therefore do not need to be considered in the simulation. The final number of parts in the simplified lock model is 13 different parts excluding screws. The parts and part names can be seen in Figure 3.2-3.5. This version of the lock is hereafter referred to as the *lock system*. The Face plate together with the Coverplate and Case are henceforth called the *lock house*. In Table 3.1 all lock parts material data used are presented.



Table 3.1: Used material data for parts in the lock system.

Part name	Material	Density, $\rho$ [kg/m <sup>3</sup> ]	Young's modulus, $E$ [GPa]	Poisson's ratio, $\nu$ [-]
Case	EN 1.0327*	7 870	210	0.29
Coverplate	EN 1.0327	7 870	210	0.29
Cylinder follower	Zamak 2 [20]	6 600	85.5	0.27
Cylinder housing	Zamak 2	6 600	85.5	0.27
Face plate	<i>classified</i>	7 900	200	0.29
Hookbolt	16NiCrS4 [21]	7 800	210	0.3
Hookbolt support	EN 1.0083 [22]	7 850	220	0.29
Latchbolt	AISI 1010 [23]	7 870	205	0.29
Latchbolt arm	EN 1.0044 [24]	7 860	200	0.29
Latchbolt guide	EN 1.0327	7 870	210	0.29
Motor holder	ABS [25]	1 040	2.62	0.35
Motor unit	-	750**	-	-
Motor bracket	EN 1.0327	7 870	210	0.29
Screws	S235JR [26]	7 800	210	0.3

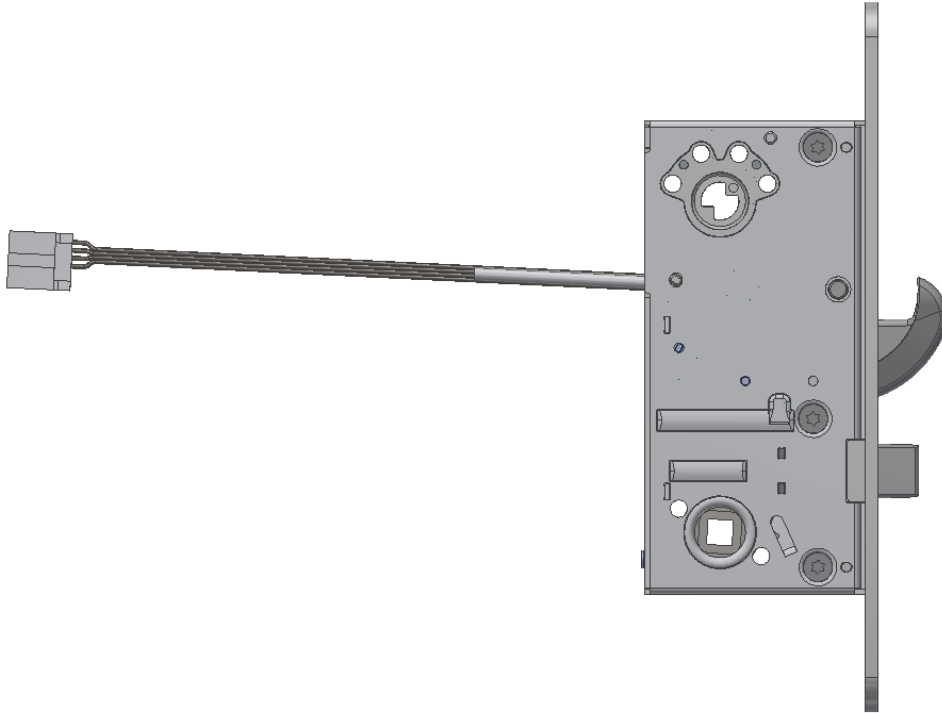


Figure 3.1: ASSA ABLOY's electromechanical lock 850C-50.

\*Material data for EN 10327 is not found but though similar materials (low carbon metals) all had the same values for the sought data (according to the material database *CES EduPack 2016*), EN 10327 is assigned these values.

\*\*Approximated density based on similar motors information found on different websites.

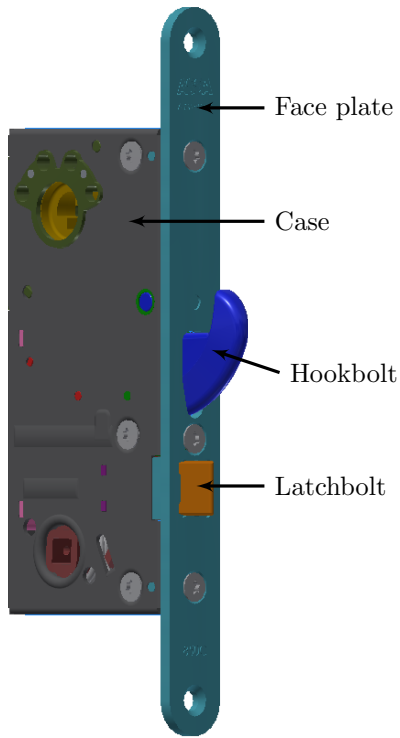


Figure 3.2: *Lock system from left side.*

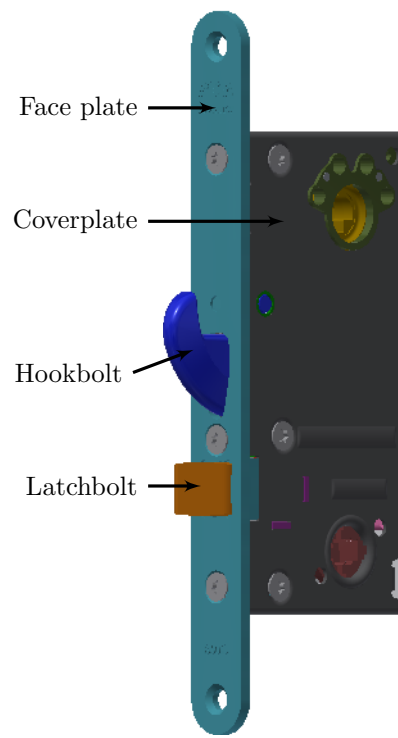


Figure 3.3: *Lock system from right side.*

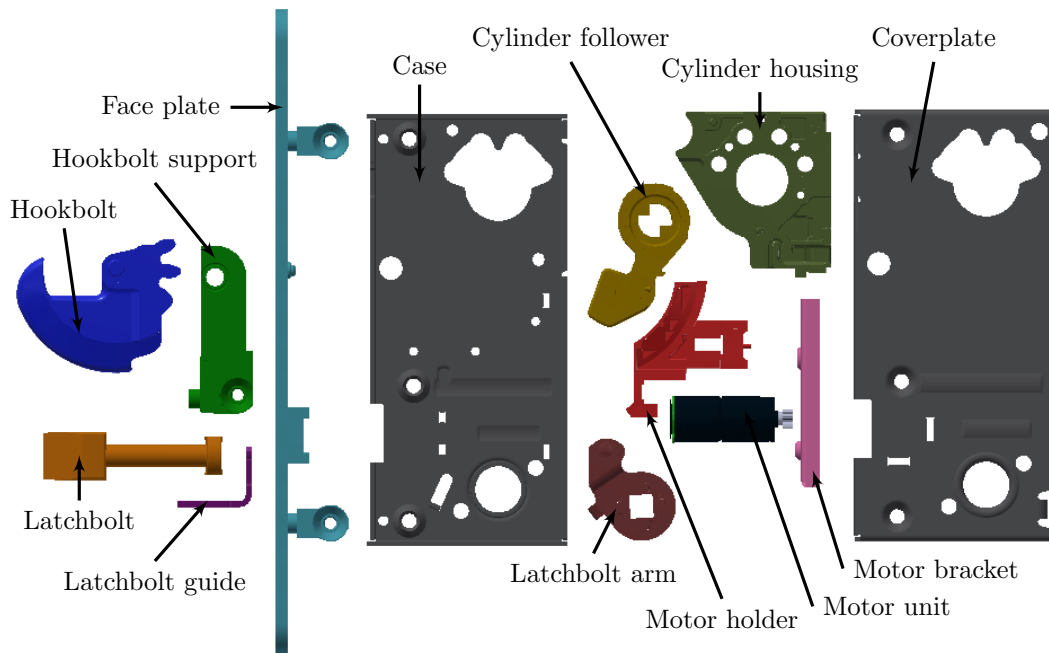


Figure 3.4: *Explode view of the lock system.*

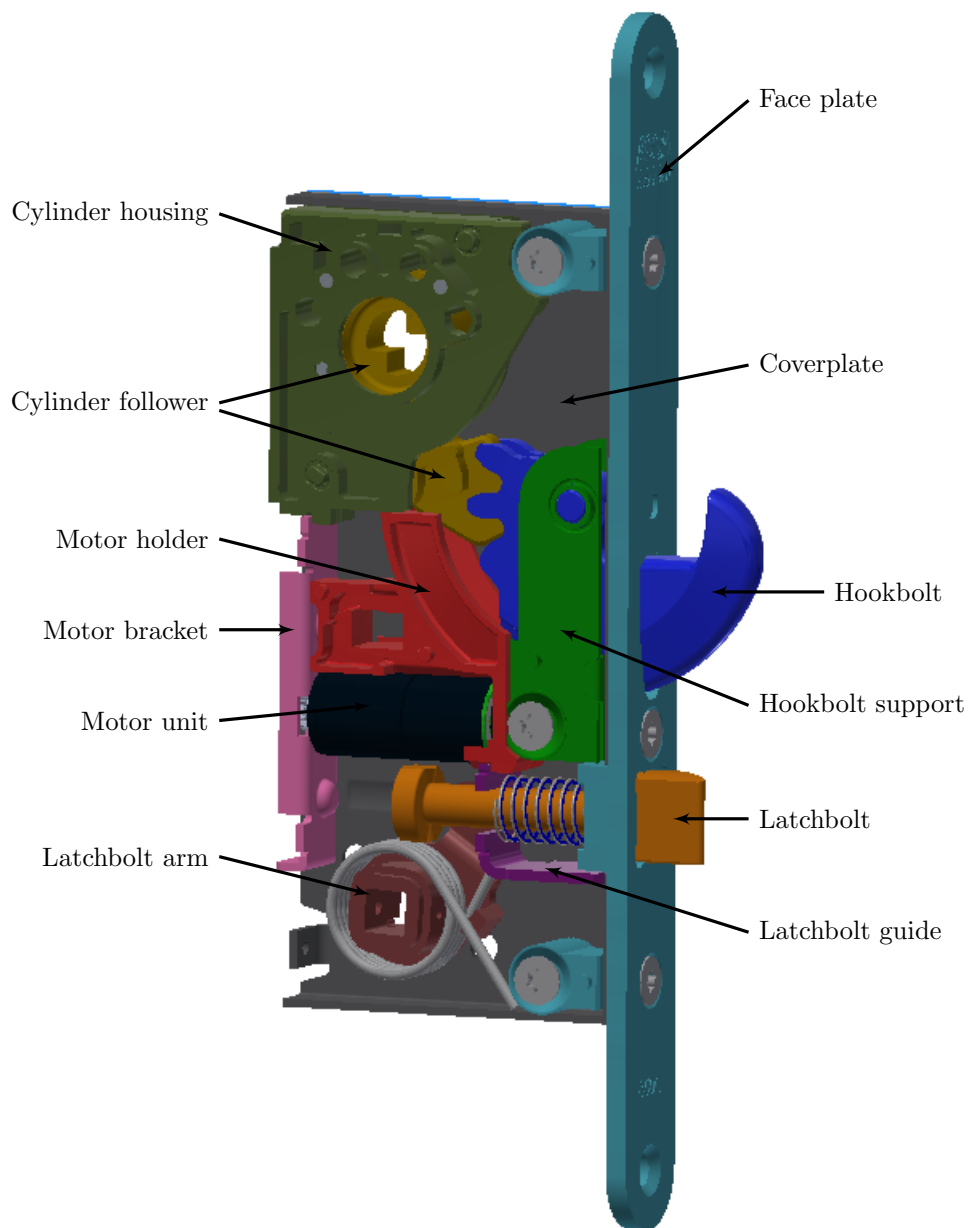


Figure 3.5: *The simplified lock system with its parts named. The Case (see Figure 3.2) is hidden to easier see the other parts.*

## 3.2 Door and door frame

The door used in these calculations is a steel door with glass windows. The door has a tree core and bent steel plate on the core's both sides, see Figures 3.6-3.7. These parts are attached together with clinching, see Figures 3.8-3.9. Figure 3.10 shows the CAD-model of the door and door frame. The material for each part and corresponding parameter values are found in Table 3.2.

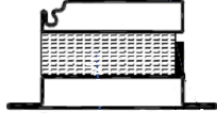


Figure 3.6: Cross section of door. Dotted area in the middle is the tree core and the black lines below and above are bent steel plates.

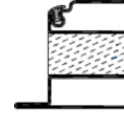


Figure 3.7: Cross section of door frame. Dotted area in the middle is the tree core and the black lines below and above are bent steel plates.

Table 3.2: Used material data for parts used in door and door frame.

Part name	Material	Density, $\rho$ [kg/m <sup>3</sup> ]	Young's modulus, $E$ [GPa]	Poisson's ratio, $\nu$ [-]
Tree core	MDF [27][28]	400	2.9	0.25
Steel plate frame	S235JR	7 800	210	0.3
Glass	glass [29]	3 330	70	0.25
Screws	S235JR	7 800	210	0.3

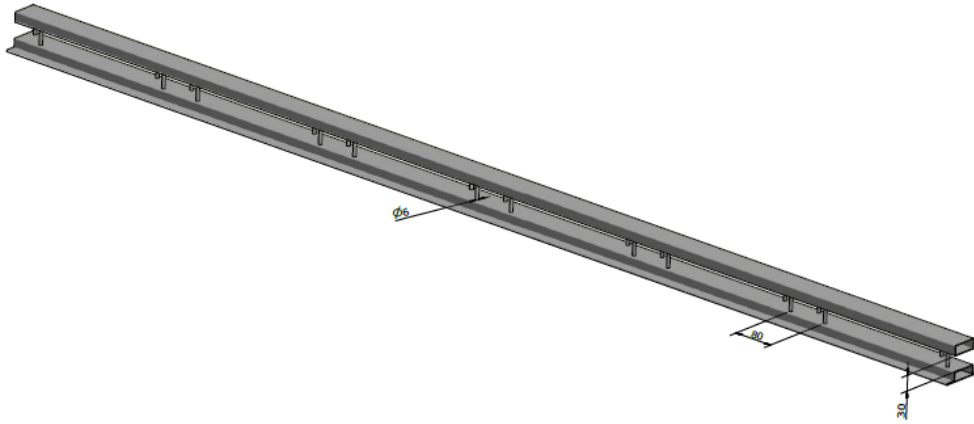


Figure 3.8: Steel frame structure for one ledge of the door with approximate positions and structure for the clinches between the steel frame's two parts. The tree core is not visible in the picture but is in the door placed between the steel frame parts with the clinching through it.

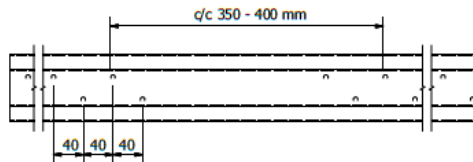


Figure 3.9: Distance in mm between clinches in the door and door frame.



Figure 3.10: *CAD-model of door and door frame. Gray parts are made of steel and blue parts are made of glass.*

## 4 Loading case

According to the standard SS-EN 1630:2011+A1:2016 [2] the door frame should be mounted into a substantial steel frame with movable steel supports. This frame should be able to hold a force of 15 kN at any of the defined points with a maximum deflection of 5 mm (see the standard for more specifics). The door frame is then mounted in tree beams in this steel frame, see Figure 4.1.



Figure 4.1: *Door mounted in tree beams in steel frame before physical test performed according to SS-EN 1630:2011+A1:2016 [2].*

The crowbar is included in the resistance class 3 and all allowed tools according to the standard can be seen in Table 4.1 and Figure 4.2. Here the loading case will be simplified and only the effect from the crowbar will be accounted for. From measured tests done by Peter Blomgren at RISE (Research Institutes of Sweden) [30] the crowbar can create applied forces on the door with a magnitude of 23-30 kN.

Table 4.1: Permitted tools according to resistance class 3 [2].

No	Tool name	Size and level
1	Screwdriver	Flat blade, length $(365 \pm 25)$ mm, blade width $(16 \pm 2.2)$ mm
1	Pipe wrench	Length $(240 \pm 20)$ mm
2	Plastic wedges	Length $(200 \pm 25)$ mm, width $(80 \pm 10)$ mm, height $(40 \pm 5)$ mm
2	Wood wedges	Length $(200 \pm 25)$ mm, width $(80 \pm 10)$ mm, height $(40 \pm 5)$ mm
1	Compass saw	With two bimetal or HSS blades, total length $(310 \pm 25)$ mm
1	Pad saw	With two bimetal or HSS blades, total length $(310 \pm 25)$ mm
1	Hacksaw	With two bimetal or HSS blades, blade length $(330 \pm 30)$ mm
1	Steel extension tube	Length $(500 \pm 5)$ mm, outer diameter $(30 \pm 0.5)$ mm, maximum wall thickness $(3 \pm 0.3)$ mm



Figure 4.2: Allowed break-in tools in resistance class 3 according to SS-EN 1630:2011+A1:2016 [2].

## 5 Method

The developing of the method for analyzing the loading case is elaborated in steps, starting with a very simplified (rigid body) model, see Section 5.1, just to set up the process between the different software. CAD-models of the door, door frame and lock system are given and those are imported as rigid bodies into Adams. Some parts of the lock system have been removed or merged as mentioned in Section 3.1. How the door and door frame are divided into sub parts is described in Section 5.1.1. The different parts in the Adams model are connected to each other with fixed joints, a contact force is created between the bolts and the door frame and a drag force is applied at the door some distance below the lock, see Section 5.1.2 for more details. This model is then implemented in HEEDS with only one run. No optimization is done here, only a first evaluation to see that the process works. The HEEDS process will be described in Section 5.4 with the convergence study part implemented.

With a working process for the first model, an improved model is developed. In this second set-up the goal is to get some information of relevance from the results, but still have the model as simple as possible. The door, door frame and lock house are therefore remade as flexible bodies instead of rigid bodies and imported into the Adams model. This model is then implemented in the HEEDS process and a new evaluation is done. Stress hot spots are output from the Adams model by the use of the *FEMData* function. To illustrate where these stresses occur a short script in MATLAB is written, these results can be seen in Section 6.1.

To validate the simulations a couple of convergence studies have been done on the mesh element size and the time steps size, see Section 5.3 for details. The whole process in HEEDS and how the different software are coupled is described in Section 5.4.

Though a visualization of the physical break-in test is of interest the Adams package MaxFlex is introduced. This introduces the possibility of nonlinear material models and therefore the material is allowed to plasticize.

The first goal is to implement a nonlinear material model on one part in the model, in this case the door frame, and analyze the advantage and disadvantage of MaxFlex. But though the knowledge about MaxFlex is limited, an even more simple model is used first, in this case the striking plate. The nonlinear material model is applied to the part and more details about the method are given in Section 5.5. When this works the nonlinear material model is applied to the door frame.

The MaxFlex tools give realistic results for the plasticizing but the simulation time is, however, rather long, much longer than with a common Finite Element Method (FEM) software. This problem, without any mechanisms, gets no advantages by using Adams MaxFlex. Therefore the model is recreated in Abaqus and most of the lock system model was provided by CAE Value. However, some parts and contacts needed to be added before the model was complete, Section 5.6.

The model of the lock system in Abaqus is verified against measured data from physical tests done by ASSA ABLOY. The lock system is exposed for a simple loading case and the deflection is measured, see Section 5.7 for further details.

## 5.1 Rigid Body model

This set-up exists of only rigid bodies which are connected to each other in Adams, see Section 5.1.1-5.1.2 for geometry description and connections in Adams. The goal with the first set-up is to create the working process to have a good base that can be developed further.

### 5.1.1 Geometry

The lock system is divided in parts as described in Section 3.1 with the addition of the Drill protection which is a steel plate placed on the outside of the Coverplate. The Drill protection is later on removed due to that it is not force bearing and therefore its impact is neglected. The door frame is in this version of the model in one part, with an average density calculated from the total mass of the tree core and steel plates. First the door is remade to only have one hole for the lock system, instead of three holes that are in the origin model. This remaking is mostly for the visual part since the strengthening of the door because of this is negligible. The door is then divided into four sub parts; *door shell* (steel plates), *door filling* (tree core), *lower glazing* and *upper glazing*. This resulted in a total of 19 parts in the Adams model.

### 5.1.2 Connections, contacts and forces

All connections, but the hinges, are fixed joints located in origin, which position can be seen in Figures 5.1-5.3. The location has no impact on the result when there is only rigid bodies and fixed joints. The hinges that connect the door to the door frame are connected with one spherical joint at the upper hinge and one inline primitive joint at the lower hinge. The spherical joint allows free rotation around a fixed point which means that all translational degrees of freedom are locked. The inline primitive joint constraints the first part so that it only can move along a straight line defined on the second part. This means that it locks the parts in two rotations (around  $x$ - and  $z$ -axis). This results in that five of six degrees of freedom are locked for the door relative the door frame. The only degree of freedom that is free is the rotation around the hinges local  $z$ -axis, so that the door can open. All connections made in the Adams model in the first set-up are listed in Table 5.1.

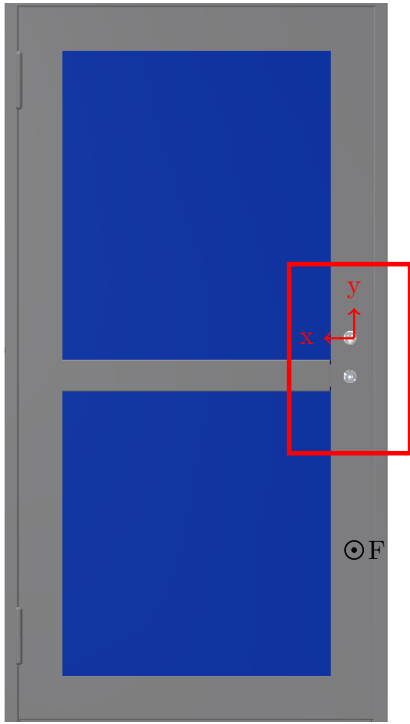


Figure 5.1: Model with zoom box for the coordinate systems position.

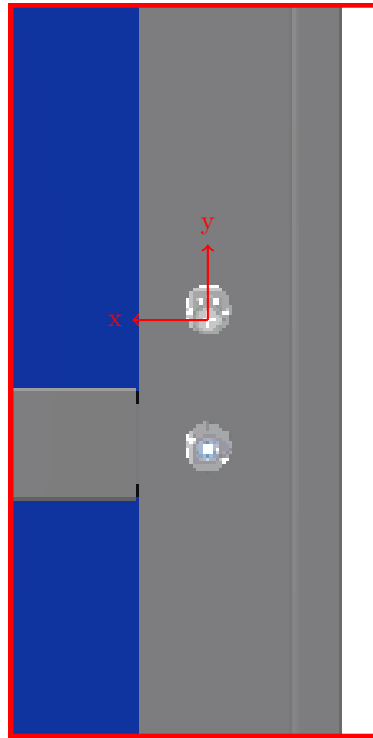


Figure 5.2: Zoom box for the coordinate systems position.

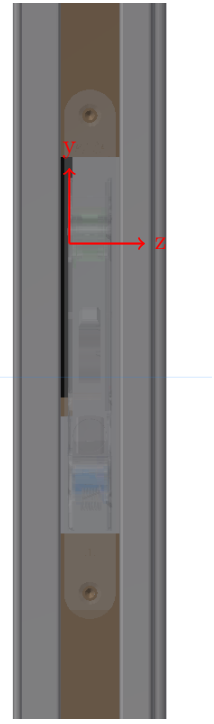


Figure 5.3: Coordinate system position from side view.

Table 5.1: Connections in Adams model in the first set-up.

Connection Type	First Body	Second Body	Comment
Fixed Joint	Door frame	Ground	Locks whole model to the ground.
Spherical Joint	Door shell	Door frame	Lower hinges connection, locking three of six degrees of freedom (all translational).
Inline Primitive Joint	Door shell	Door frame	Upper hinges connection, locking two of six degrees of freedom (two rotational).
Fixed Joint	Door filling	Door shell	Keeps the door as one part.
Fixed Joint	Lower glazing	Door shell	
Fixed Joint	Upper glazing	Door shell	
Fixed Joint	Case	Door shell	
Fixed Joint	Coverplate	Case	
Fixed Joint	Drill protection	Coverplate	
Fixed Joint	Face plate	Coverplate	
Fixed Joint	Hookbolt support	Face plate	
Fixed Joint	Hookbolt	Hookbolt support	
Fixed Joint	Motor holder	Hookbolt support	
Fixed Joint	Motor unit	Motor holder	
Fixed Joint	Motor bracket	Motor unit	
Fixed Joint	Latchbolt guide	Motor holder	
Fixed Joint	Latchbolt	Latchbolt guide	
Fixed Joint	Cylinder housing	Face plate	
Fixed Joint	Cylinder follower	Cylinder housing	
Fixed Joint	Latchbolt arm	Latchbolt guide	



The implemented contacts are between the bolts (Hookbolt and Latchbolt) and the door frame. To create the contacts some settings are defined; a material *stiffness* that is used to calculate the normal force for the impact model, *damping constant* that defines the damping properties for the contacting material, *penetration depth* at which depth Adams turns on full damping and a *force exponent* that is used to calculate the normal force according: (  $\text{stiffness} \cdot \text{penalty}^{\text{exponent}}$  ). The values of the settings can be seen in Table 5.2. This is to hold the door closed, even when a drag force is applied. The drag force is applied on the door at location  $(x,y,z)=(-0.6,0,0)$  m, see point F in Figure 5.1, with a magnitude of 2 300 N in negative  $z$ -direction. The exact location and magnitude of the force in this model is not of importance though the model is only created to set up the process.

Table 5.2: Settings for the contact force.

Stiffness	2000 MN/m
Force Exponent	2.2
Damping	1 kN·sec/m
Penetration Depth	$1 \cdot 10^{-5}$ m

## 5.2 Mixed flexible and rigid body model

In the second set-up the model is developed one step further and some flexible bodies are introduced. The door, door frame and lock house are developed to flexible bodies in ANSA, remade to Modal Neutral Files (MNF) and then implemented in Adams. The remaking of to MNFs is done by exporting the ANSA-file as a Nastran-file and then running a Bulk Data File (BDF) in Nastran which generates the MNF, see Appendix A for BDF used here. More detailed descriptions about the meshing, connections and other settings can be found in Sections 5.2.2-5.2.3. From this set-up more realistic results are presented and an evaluation of how realistic the models is could be done, see Section 6.1.

### 5.2.1 Geometry

When remaking the parts to flexible bodies, some parts in Adams are merged. The Face plate, Coverplate and Case together creates the flexible body *lock house*. The *door* is also divided into more than one part in the first set-up, but here these parts only are separated in the ANSA model for the possibility to have different materials. The *door frame* is kept as one part. The Drill protection is from hereon removed due to that its strengthening is negligible. This results in 13 parts in the Adams model.

### 5.2.2 Clean up and meshing of flexible bodies

The meshing of the flexible bodies are made in ANSA and all parts are first “cleaned up” in the program. Small errors that occurred when importing the geometry are corrected and surfaces with very small areas are merged to simplify the model. Most of the parts are modelled as shell elements by first creating a mid surface of the geometry. All screws are modelled as beam elements, this neglects pre-stresses, friction forces and contact forces but these are in this stage neglected due to that they are not expected to have any significant impact on the result.

The tree core in the door and the tree core and striking plate in the door frame are modeled by solid elements, while all the other parts (the glass and steel plates) are modelled by shell elements. The steel plates are 1.5-5 mm thick and the glass is approximate 15 mm thick, therefore a simplification with shell elements are used. This can be done due to the small dimensions compared to the geometries height and width. The glass is fasten to the door with 80 small rigid body elements that go between one node on the glass and one on the door (also called RBE2). An example of one of those can be seen in Figures 5.4-5.6. The clinching between the tree core and the steel frames are modelled as beam elements. The contacts for the beam elements are spread over some few nodes (with RBE2) to create a more accurate model, see Figure 5.6.

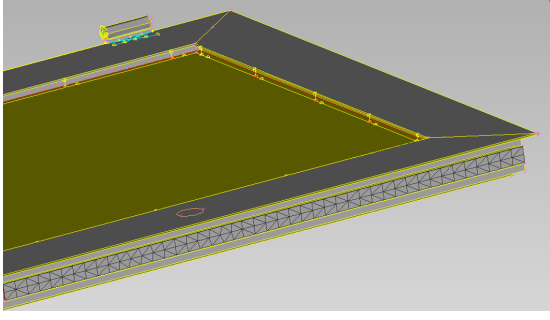


Figure 5.4: Upper part of door, gray parts are steel and tree frame while the dark yellow part is the glass. Small yellow lines between the frame and the glass are RBE2:s.

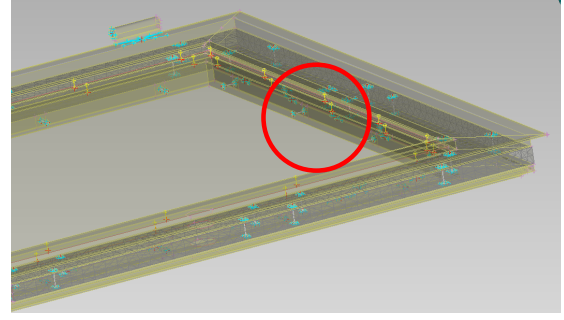


Figure 5.5: Same view as in Figure 5.4 but with all RBE2 highlighted. The part inside the red circle can be seen enlarged in Figure 5.6.

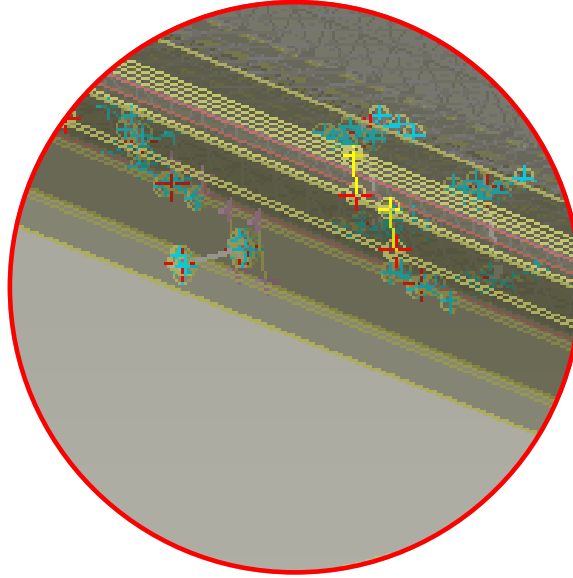


Figure 5.6: Zoomed part from Figure 5.5. Yellow lines between the red and yellow cross are RBE2:s that connects the glass to the door. The blue crosses are RBE2:s that connect the beam elements that clinch the steel and tree together.

The lock house is modelled by mostly shell elements but with some beam elements for a couple of screw connections and a small connection distance, see Figure 5.7. The mesh size is based on default mesh with some refinements at sharp edges and other small details. The total number of elements for the three parts (door, door frame and lock house) are 171 338, and the total numbers of shell, solid respectively beam elements are found in Table 5.3.

Table 5.3: Number of elements in the different parts.

	Solid elements	Shell elements	Beam elements	Total
Door	23 041	32 119	264	55 424
Door frame	59 497	28 461	166	88 124
Lock house	19 682	8 093	15	27 790
Total	102 220	68 673	445	171 338

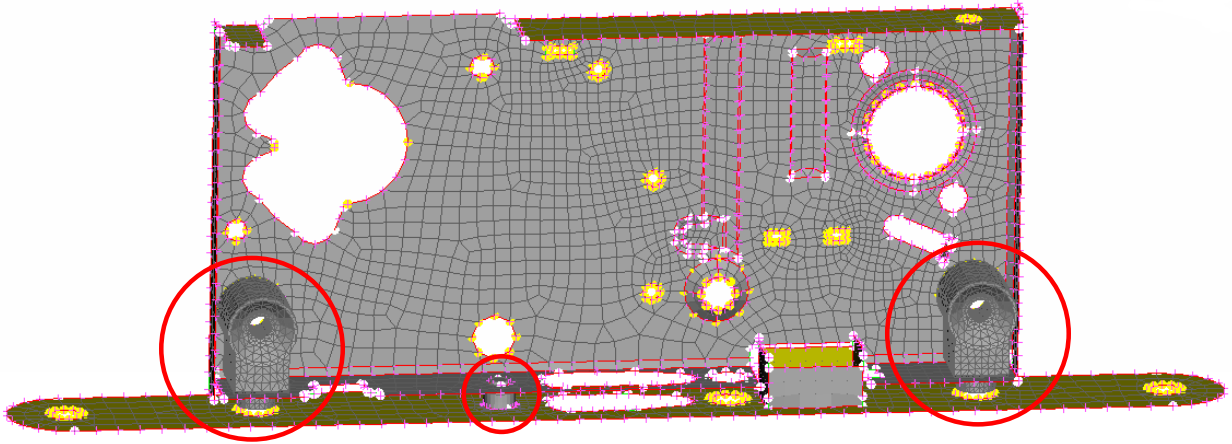


Figure 5.7: *Mesh of lock house without the Coverplate. Parts made by solid elements are hedged by red circles.*

### 5.2.3 Adams model

With flexible bodies involved the locations for the fixed connections become important. A flexible body can also have more than one fixed constraint without being overconstrained. The door frame is fastened to the ground with fixed constraints at eight screw holes at its sides, see Figure 5.8. The door's connection to the door frame is the same as in the first set-up, see Section 5.1.2. All connections are stated in Table 5.4 and with its position marked in Figure 5.8-5.11.

The contact force between the Hookbolt and door frame is still there to prevent the door from opening. But the contact force between the Latchbolt and door frame is removed due to that this bolt is easily pressed in during a real break-in and therefore does not prevent the door from open. To prevent unnatural movement contact forces are also implemented between the door and door frame and between the lock house and door. Settings for the contact force is the same as in the first set-up, see Section 5.1.2. The drag force is applied in the same location as before,  $(x, y, z) = (0, -0.6, 0)$  m, but is this time ramped up to a maximum magnitude of 23 kN over a time of 2.5 seconds. The door and door frame are designed to be critically damped to avoid oscillating vibrations. In Adams the linear (MNF-based) flexible body damping is done on a modal basis since those flexible bodies are essentially a superposition of individual modes [19].

Table 5.4: Connections in the mixed flexible and rigid body model.

ID	Connection Type	First Body	Second Body	No.*	Comment
1	Fixed Joint	Door frame	Ground	8	Locks whole model to the ground where the door frame has screw holes.
2	Spherical Joint	Door	Door frame	1	Upper hinges connection, locking three of six degrees of freedom (all translational).
3	Inline Primitive Joint	Door shell	Door frame	1	Lower hinges connection, locking two of six degrees of freedom (two rotational).
4	Fixed Joint	Face plate	Door	2	Locks the lock house to the door with two screw holes in the Face plate.
5	Fixed Joint	Face plate	Hookbolt support	1	
6	Fixed Joint	Lock house	Hookbolt support	1	Fixed joints at screw holes through lock house into the Hookbolt support.
7	Fixed Joint	Lock house	Hookbolt support	1	
8	Fixed Joint	Case	Hookbolt support	1	
9	Fixed Joint	Lock house	Latchbolt arm	1	
10	Fixed Joint	Lock house	Cylinder housing	1	
11	Fixed Joint	Case	Cylinder housing	2	
12	Fixed Joint	Case	Latchbolt guide	2	
13	Fixed Joint	Coverplate	Latchbolt guide	2	
14	Fixed Joint	Case	Motor holder	2	
15	Fixed Joint	Case	Motor bracket	2	
16	Fixed Joint	Latchbolt	Latchbolt arm	1	
17	Fixed Joint	Hookbolt	Hookbolt support	1	
18	Fixed Joint	Cylinder follower	Cylinder housing	1	
19	Fixed Joint	Motor unit	Motor holder	1	

---

\*Number of similar joints but at different locations.

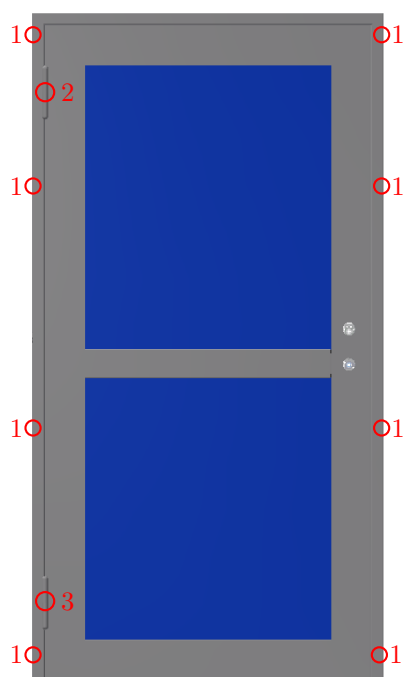


Figure 5.8: Door with connection positions marked with red circles.

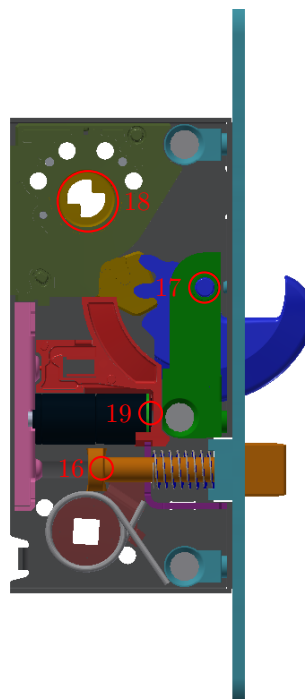


Figure 5.9: Lock system with connection position marked with red circles.

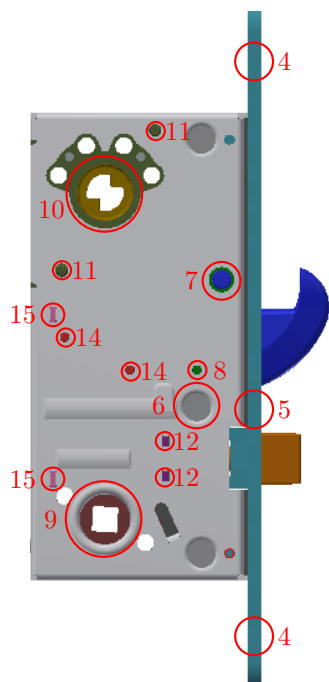


Figure 5.10: Lock system from left side with connection positions marked with red circles.

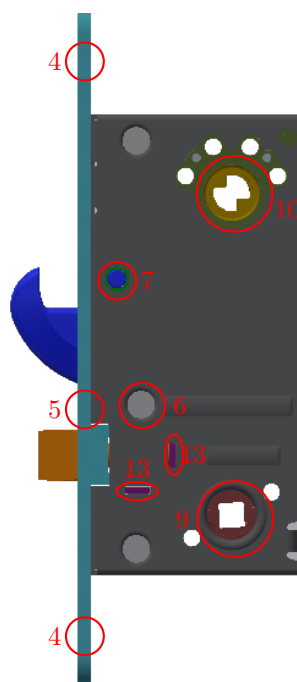


Figure 5.11: Lock system from right side with connection positions marked with red circles.

### 5.3 Convergence studies

To validate the results, some convergence studies of the mesh size and time steps size are done. This is made with the help of the optimization software HEEDS where some design variables are defined to make a process that creates multiple runs and presents the results. The step sizes are defined by the number of time steps and can easily be referred to in HEEDS though it is already input settings. It is defined as a design variable and after some preliminary elaborations it was set to vary between 50 and 1000 number of steps, which with a simulation time of one second results in time steps sizes of 1 to 20 ms. The number of time steps (or step size) in the convergence study is based on six number of evaluations. The different evaluations done are seen in Table 5.5. The results are then presented in Section 6.2.1.

Table 5.5: Convergence study evaluations of what number of time steps that are used.

Evaluation number	1	2	3	4	5	6
Number of time steps	50	100	200	400	800	1000
Step size [ms]	20	10	5	2.5	1.25	1

A convergence study of the mesh size is set-up in HEEDS, with the purpose of evaluating a refined mesh on all flexible bodies in each evaluation. To make the mesh size as a design variable in HEEDS, remaking of the ANSA models (of the flexible bodies) and loading of the MNFs are necessary. The mesh is here based on a simple and rather coarse mesh. In each evaluation, the whole mesh is refined by the mesh refinement function in ANSA and therefore the mesh over the whole geometry will be finer in each evaluation. The mesh refinements convergence study is set-up for five different evaluations, the first one with the original coarse mesh and then with an increasing number of refinements from one to four. This process is described in Section 5.4.

Due to gained knowledge during the process these evaluations were never ran though the mesh convergence study is better done on each flexible body by itself. A simpler HEEDS process is instead set up for only the lock house, the most interesting of the flexible bodies. A new Adams-file is created for this, where the lock house is locked in its screw holes as before and a force is acting on some nodes around the Hookbolt's hole in the Face plate, see Figure 5.12. Approximate number of elements on the lock house in each evaluation is shown in Table 5.6.

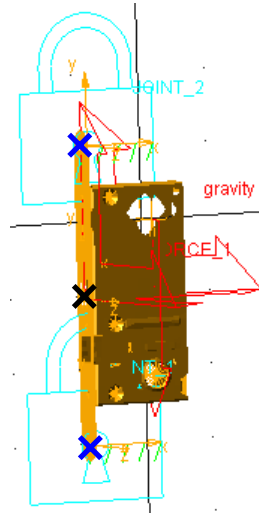


Figure 5.12: Lock house model in Adams with blue lock indicating fixed points (marked with blue crosses) and the red three-pointed arrow indicating the force (which is applied at the black cross).

Table 5.6: Convergence study evaluations of what approximate number of elements that are used.

Evaluation number	1	2	3	4	5
Refinements	0	1	2	3	4
Number of elements	15 000	90 000	460 000	2 220 000	2 530 000

## 5.4 HEEDS structure

HEEDS is built up by boxes that have their own functions, see squares in Figure 5.13. All boxes have specified input and output data and hence links the boxes together due to that one's output could be another one's input. The three boxes to the right (box number 6-8) form the base structure created according to Section 5.1. To implement the automatic mesh refinement in HEEDS multiple new boxes in the software are needed. Those are implemented to the left of the first ones due to that their function should happen before the actual evaluation. For the convergence study on time step size only the boxes 6-8 are needed. All boxes' functions and their input and output files are presented in the list below and a simplified flow chart can be seen in Figure 5.14.

For the actual convergence study ran on mesh size of the lock house the boxes 2 and 3 are removed from the HEEDS process. Otherwise the process in HEEDS is exactly the same, even though another Adams-file in the simulation-box is ran.

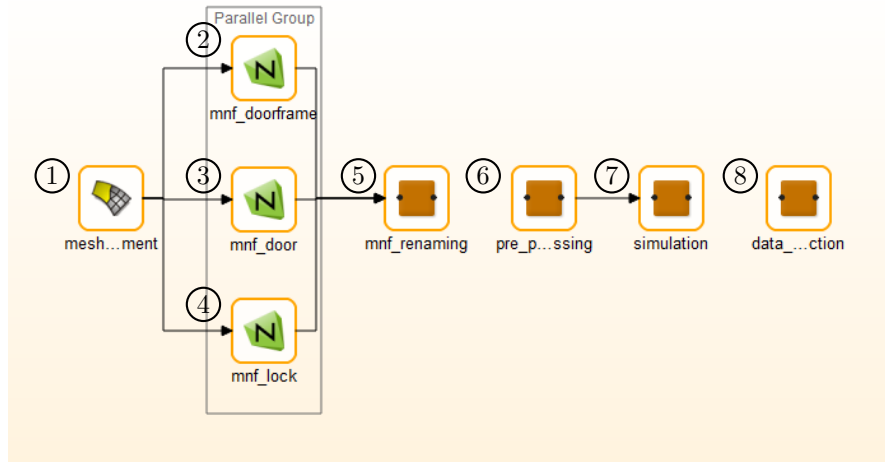


Figure 5.13: Picture of the process in HEEDS Graphical User Interface (GUI) with all boxes labeled with numbers.

1. ANSA-box: Opens ANSA and reads an ANSA-script that refines the mesh of the flexible bodies (door, door frame and lock house) a specified number of times.
  - **Input:** ANSA-script
  - **Output:** Three Nastran-files, one for each flexible body
2. Nastran-box (door): Opens Nastran and runs a BDF (see Appendix A) including the output Nastran-file of the door from box one. This generates a MNF of the part.
  - **Input:** BDF and Nastran-file for the door
  - **Output:** MNF of the door
3. Nastran-box (door frame): Opens Nastran and runs a BDF (see Appendix A) including the output Nastran-file of the door frame from box one. This generates a MNF of the part.
  - **Input:** BDF and Nastran-file for the door frame
  - **Output:** MNF of the door frame
4. Nastran-box (lock house): Opens Nastran and runs a BDF (see Appendix A) including the output Nastran-file of the lock house from box one. This generates a MNF of the part.
  - **Input:** BDF and Nastran-file for the lock house
  - **Output:** MNF of the lock house
5. MNF-renaming-box: Runs Python script that moves and renames the MNFs to the name and folder used by Adams.

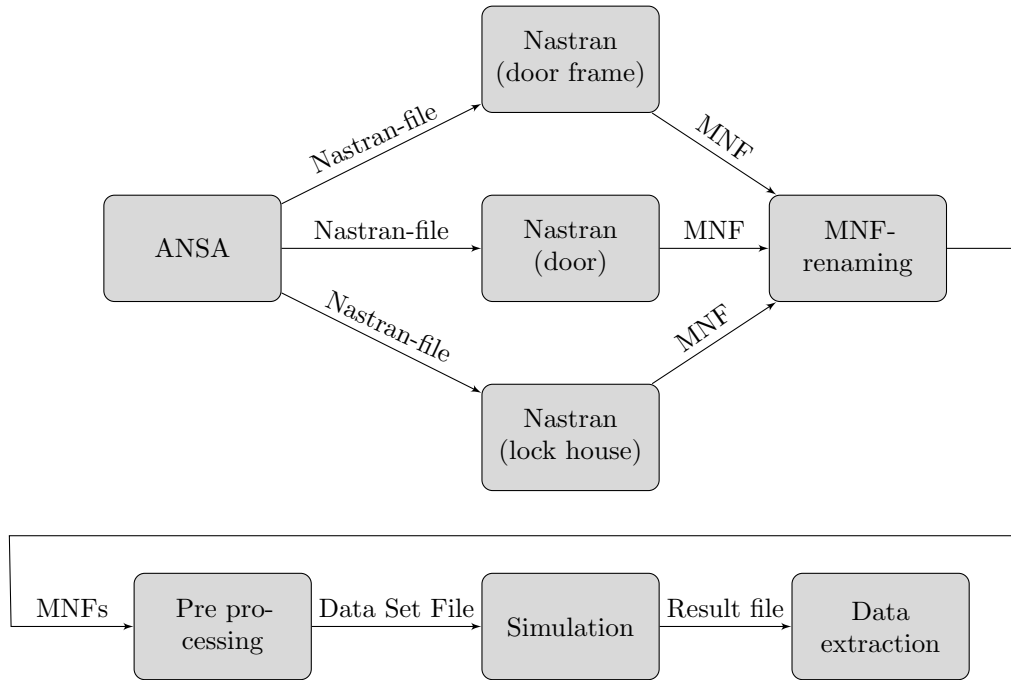


Figure 5.14: *Simplified flow chart of HEEDS structure.*

- **Input:** MNFs of the parts from box 2-4
  - **Output:** MNFs of the parts saved with another name in the working folder for Adams folder
6. Pre-processing-box: Read the Adams database including the model and creates simulation files.
- **Input:** Command file that reads the Adams database of the model and changes some force parameters
  - **Output:** Adams data set file including the model and a parasolid file
7. Simulation-box: Reads the simulation script including total simulation time and number of time steps (see Appendix B). Running the simulation and gives output about von Mises hot spot stresses in the flexible bodies.
- **Input:** A simulation script, Adams data set file, a parasolid file and matrix files\* for each flexible body
  - **Output:** A simulation running message and a document including top 15 maximum hot spot stresses in each flexible body.
8. Data extraction-box: Reads the simulation results and saves the contact force between the Hookbolt and door frame over time into a text file.
- **Input:** Script that saves chosen results from the simulation into a text file.
  - **Output:** Text file with specified results.

---

\*The matrix file contains inertia invariants and the portion of the flexible body modes shapes that correspond to marker locations.



## 5.5 MaxFlex model

To learn about the MaxFlex tool in Adams a simple model is chosen in the first step. This is also time saving though a smaller and simpler model need less simulation time. The striking plate, usually mounted on the door frame, is used though it is an existing but small part. For further details see Section 5.5.1. When this model is running and some knowledge about the MaxFlex tool is gained the nonlinear material model is implemented on the whole door frame, see Section 5.5.2.

### 5.5.1 Striking plate

The striking plate is remodelled by shell elements for simplifications. The material data for this part is redefined in ANSA to a nonlinear material, see Figure 5.15. This is done by activating the material parameter MATEP in ANSA and assign it SOL400 based on a created table. The model information is imported in an BDF together with some output information and parameter settings, see Appendix C for parameters used in the BDF. This file is then imported in a new Adams database.

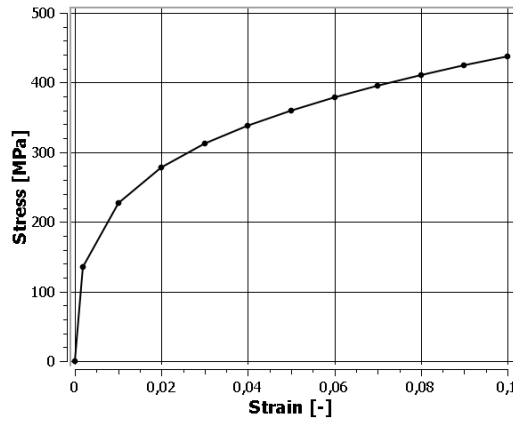


Figure 5.15: *Stress-strain curve for material S235JR [31].*

In Adams the striking plate is locked with fixed joints in its screw holes and a force is applied on one side of the inwards fold, see the black cross in Figure 5.16. The applied force is defined as a step function over a time of 0.6 seconds with a maximum magnitude of 650 N, `STEP(TIME,0,0,.6,-650)`. The force is applied in the opposite direction of the red force arrow in the figure. A time step of 0.8 ms is used. Deformation, stress and plastic strain are set to be output data from the simulation.

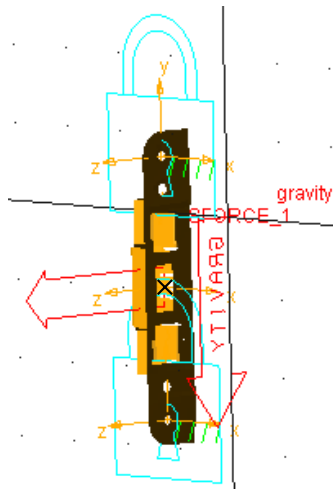


Figure 5.16: *Striking plate from Adams model with fixed joints (blue locks) and applied force (red arrow pointing to the left) with its attached point in the black cross.*

### 5.5.2 Door frame

The material in the door frame model is changed to a nonlinear material in the same way as for the striking plate, see Section 5.5.1. The same material data is used. Due to a more complex model some settings in the BDF are changed to decrease the running time somewhat, see Appendix D for whole BDF. The stress output is only saved for the steel frame and not the tree core.

In Adams the door is locked to its position by fixed joints at seven of its eight screw holes and the force is applied at the eighth hole, see Figures 5.17-5.19. The applied force is defined as a step function over a time of one seconds with a maximum magnitude of 23 kN,  $\text{STEP}(\text{TIME}, 0, 0, 1, -23000)$ . A step size of  $4.55 \cdot 10^{-4}$  seconds is used. Simulation settings used are presented in Table 5.7.

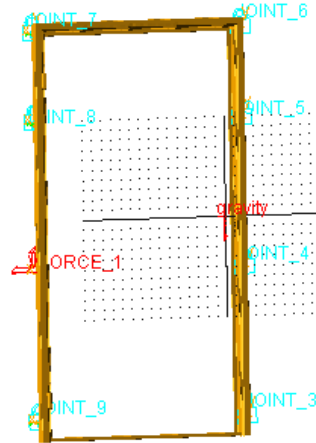


Figure 5.17: *MaxFlex* model of the door frame in Adams. Fixed joints are marked by blue locks and the red arrow on the left side symbolizes the applied force.

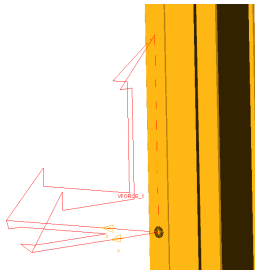


Figure 5.18: *MaxFlex* model of the door frame, zoomed in to where the force is applied at a RBE2 in a screw hole.

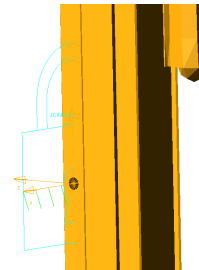


Figure 5.19: *MaxFlex* model of the door frame, zoomed in to where one of the fixed joints is applied at a RBE2 in a screw hole.

Table 5.7: Solver settings for MaxFlex model of door frame.

Method used for integrating the differential equations of motion:	GSTIFF*
Technique used for equations describing mechanical system:	I3**
Relative and absolute local integration tolerance:	$10^{-3}$
Maximum integrator time step allowed:	None
Evaluates the Jacobian at every iteration:	TFFF TFFF TFFF TF*** (default for dynamic simulations)

## 5.6 Abaqus model

A model of the lock system in Abaqus is provided by CAE Value. The model contains most of the parts included in the simplified model of the lock system but with none of the parts merged as in the Adams model. The parts are instead fixed to each other with tie contact in Abaqus (with a position tolerance of 0.05 mm). The Latchbolt, Latchbolt arm and Motor are not yet included, see Figures 5.20-5.21. All parts that are in contact are fixed with the tie contact or with a RBE2. To prevent the Hookbolt from penetrate the parts around it when loaded, contact pairs are created between the Hookbolt and the Face plate, Hookbolt support, Motor holder, Case and Coverplate.

The model is updated with the Latchbolt and the Latchbolt arm, see Figure 5.22. Nonlinear materials with isotropic hardening are added to the Face plate, the Coverplate, the Case, the Latchbolt guide and the Motor bracket. The materials stress-strain curves are presented in Figures 5.23-5.24. Remaining parts in the model are given the material data stated in Table 3.1. Tie contacts are used between the Latchbolt and the Latchbolt arm, the Latchbolt arm and the Case and between the Latchbolt arm and the Coverplate. To prevent the Latchbolt from penetrating the lock house contact pairs are created between the Latchbolt and the Face plate and between the Latchbolt and the Case.

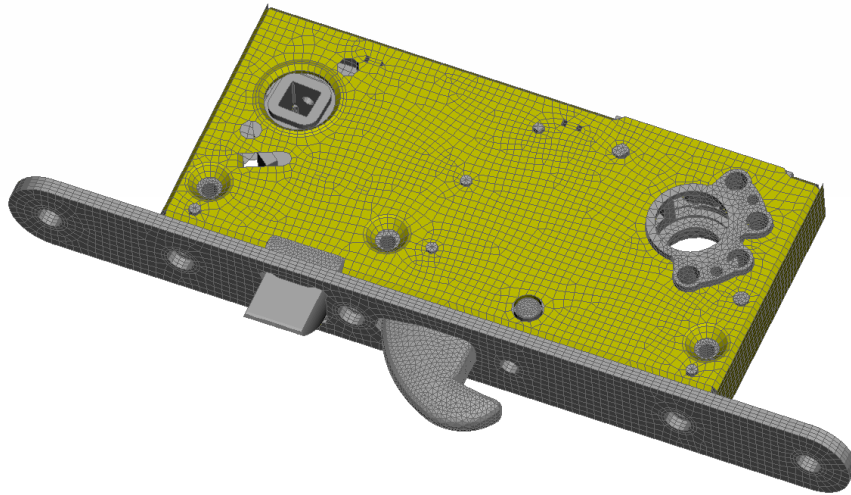


Figure 5.20: *FE model of the lock system.*

\*Integrator that uses backward differentiation formulas and fixed coefficients for prediction and correction.

\*\*Equation formulation that ensures that the solution satisfies all constraints, does not ensure that the velocities and accelerations calculated satisfy all first- and second-time derivatives and monitors integration error only in system displacements, not in velocities. It is fast, but the Jacobian matrix can become ill-conditioned at small step sizes.

\*\*\*A ten character strings that together establish the pattern for evaluating the Jacobian matrix during the modified Newton-Raphson solution for a dynamic simulation. For each iteration, T or TRUE indicates that Adams Solver is to evaluate the Jacobian, and F or FALSE indicates that Adams Solver is not to evaluate the Jacobian, instead it is to use the previously calculated Jacobian matrix as an approximation of the current one.

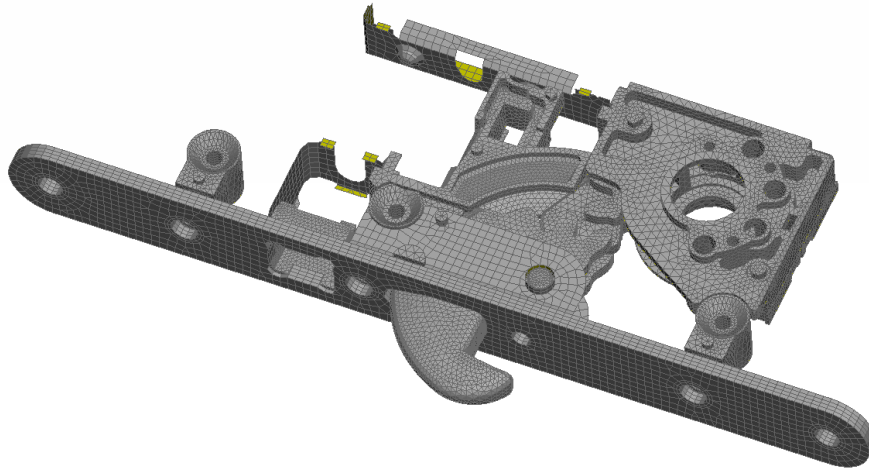


Figure 5.21: *FE model of the lock system without the Coverplate and the Case.*

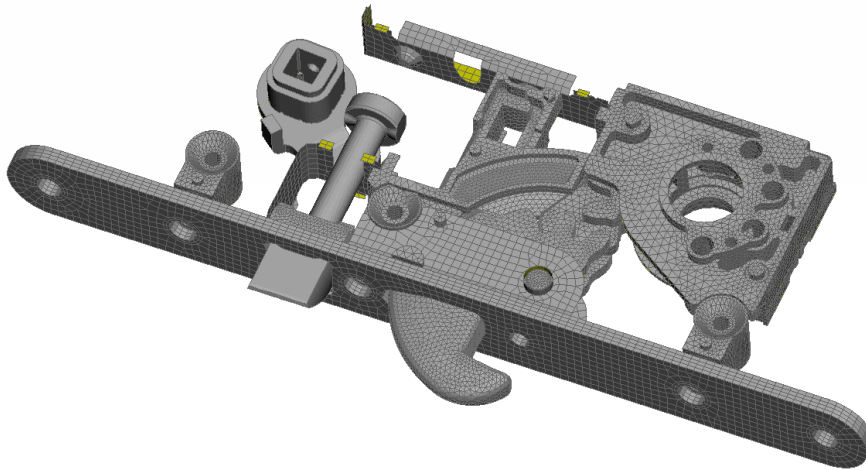


Figure 5.22: *Developed model of the lock system without the Coverplate and the Case.*

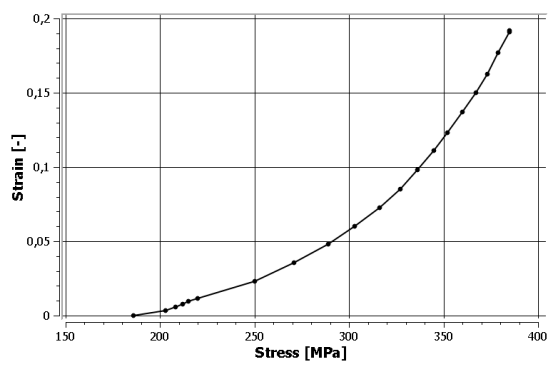


Figure 5.23: *Stress-strain curve of the plastic zone for the nonlinear material EN 1.0327, applied on the Case, Coverplate, Latchbolt guide and Motor bracket [31]. Yield limit is 186 MPa.*

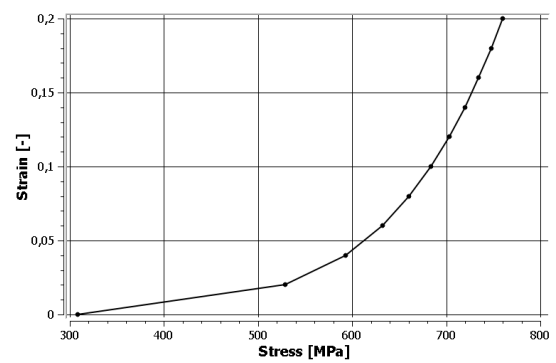


Figure 5.24: *Stress-strain curve of the plastic zone for the nonlinear material applied to the Face plate [31]. Yield limit is 308 MPa.*

## 5.7 Verification of Abaqus model of the lock system

To verify the Abaqus model of the lock system a standardized loading case is compared with measured data from a physical test done by ASSA ABLOY, see picture from testing in Figure 5.25. Details about the measured data are presented in Section 5.7.1. In the physical test the lock system is positioned horizontally in a mounting stand, see illustration in Figures 5.26-5.27. The mounting stand locks the screws in all degrees of freedoms but in the  $y$ -direction. This is to not restrain the bending when the force is applied. The force is applied at the Hookbolt a distance of three millimeter from the Face plate. To prevent the lock system from tipping because of the force a beam is fastened above the upper side of the lock system.

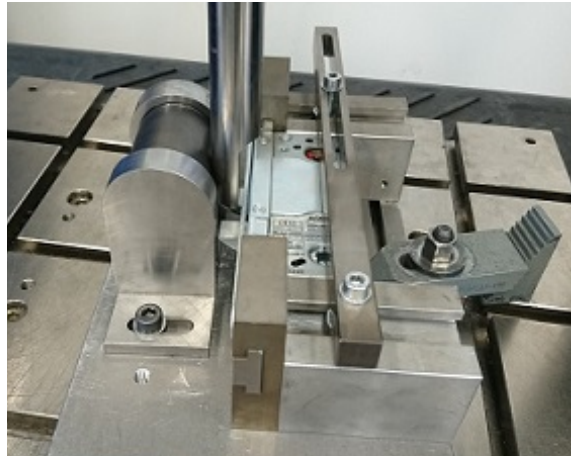


Figure 5.25: Lock system locked in the mounting stand at the physical loading case.

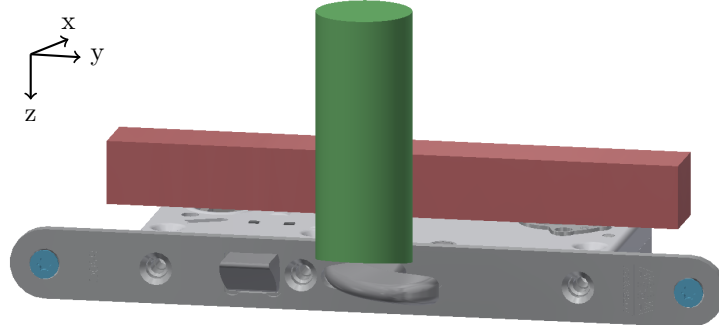


Figure 5.26: Simplified model of the loading case. The blue screws hold the lock fixed in all degrees of freedoms but in the  $y$ -direction, the red beam blocks the lock's back from tipping upwards and the green cylinder is the steel part that applies the force.

The loading case is then applied to the Abaqus model as described here. In each screw hole, a rigid body is connected and its center node is fixed in  $x$ - and  $z$ -directions and in all rotational degrees of freedom. A created stiff rigid body element, fixed in its middle, symbolizes the beam that restrains the lock system from tipping, see Figure 5.28. Four forces each applied at a couple of nodes on the Hookbolt are applied to distribute the force without stiffen the Hookbolt too much. This is done by creating four RBE2:s at the Hookbolt, see Figures 5.29-5.30, and applying the forces at each RBE2's center node. The total force magnitude is the same as at the physical test, the force over time is plotted in Figure 5.31.

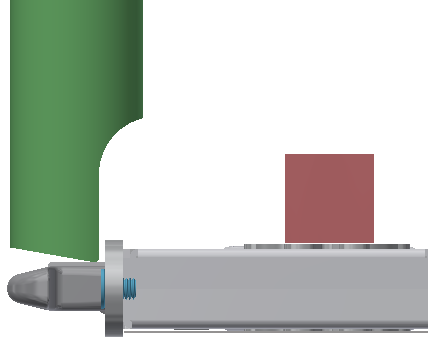


Figure 5.27: *Simplified model of the loading case. The blue screws hold the lock fixed in all degrees of freedoms but in the y-direction, the red beam blocks the lock from tipping forward and the green part is the steel part that applies the force.*

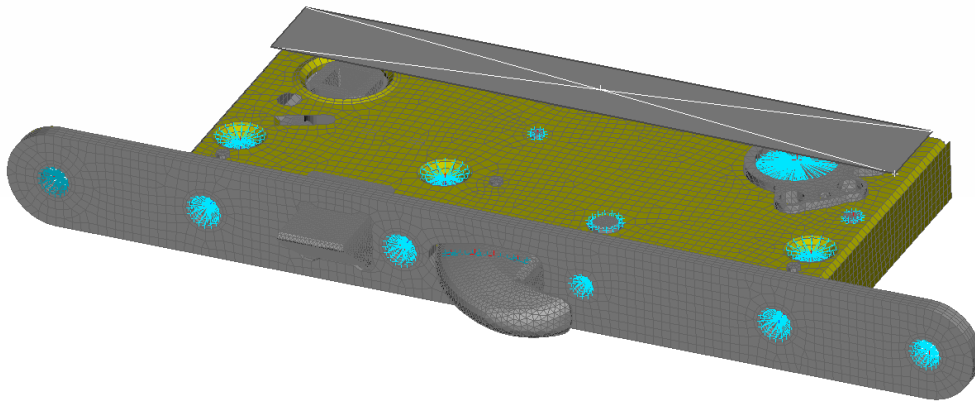


Figure 5.28: *Lock system in ANSA GUI (with Abaqus package active). Blue spiders are RBE2:s connecting the different parts to each other or spreading the force/joint on multiple nodes. The grey part (with a big white cross) on the locks upper back side is the stiff rigid body element.*

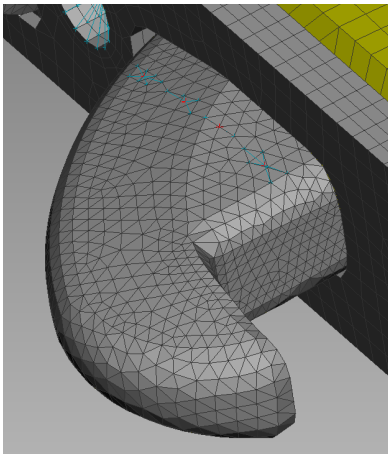


Figure 5.29: *The figure illustrates how the force is applied to the Hookbolt in Abaqus. The blue spiders (partly hidden) are RBE2:s and the force is applied at its center nodes, the small red dots in the spiders.*

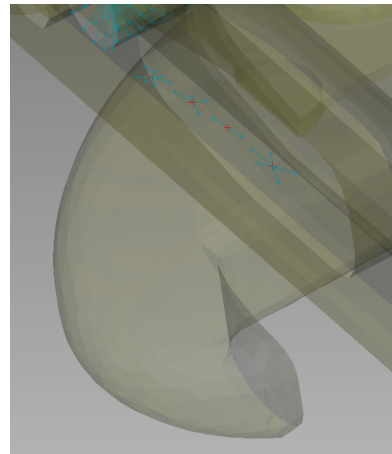


Figure 5.30: *The figure illustrates how the force is applied to the Hookbolt in Abaqus. The blue spiders are RBE2:s and the force is applied at its center nodes, the small red crosses in the spiders. All parts but the spiders are transparent for easier visibility.*

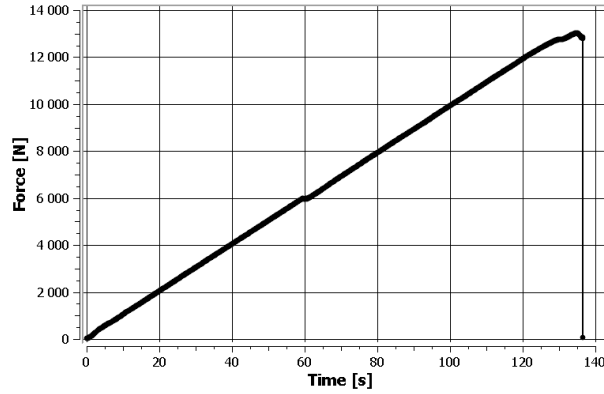


Figure 5.31: *Force versus time measurements from physical tests.*

The simulation ran for 136 seconds, to the time when the force is the highest, with automatic step size settings. The first step is 2 seconds, minimum step size allowed is 0.001 seconds and maximum step size is 3 seconds.

To evaluate how high impact the nonlinear material data has on the result some simulation with altered data are done as well. The stress values for the two different nonlinear materials (the Face plate material and EN 1.0327) are translated by approximate 5 % of its original yield limit, both in negative and positive direction, see Evaluation 2-7 in Table 5.8. Evaluation 8-10 is with the stress translated 20 %, 30 % and 50 % in the negative direction.

Table 5.8: Translating stress status in MPa for altered material data in each evaluation.

Evaluation	1	2	3	4	5	6	7	8	9	10
Face plate material	-	-30	-	-30	+30	-	+30	-62	-92	-154
EN 1.0327	-	-	-20	-20	-	+20	+20	-37	-55	-93

### 5.7.1 Measure data

The physical tests performed by ASSA ABLOY give information about the load, time and movement for the crosshead that applies the force on the Hookbolt. Two similar tests are done and the data is plotted in Figures 5.32-5.33. Though the data for Test 2 is smoother, and therefore seems more realistic, this will be used to validate the model.

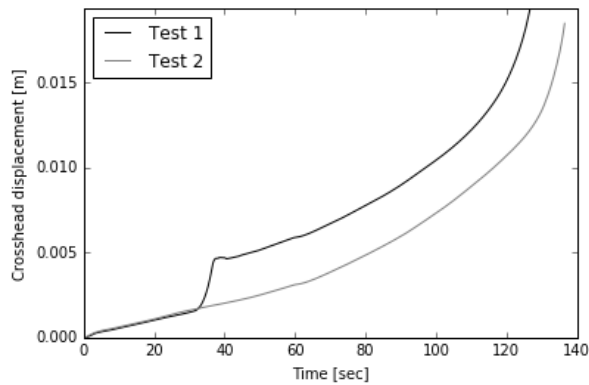


Figure 5.32: *Measure data from physical test done by ASSA ABLOY.*

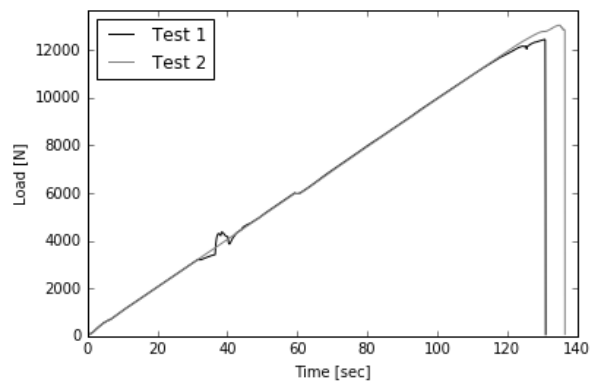


Figure 5.33: *Measure data from physical test done by ASSA ABLOY.*

## 6 Results

### 6.1 Mixed flexible and rigid model

From the second set-up with a mixed flexible and rigid body some simple results are extracted. The model is still very simplified and the force's magnitude and location are applied without further elaborations or analyses. But with a maximum force magnitude of 23 kN at a location of (0, -600, 0) mm, 50 stress hot points are analyzed, see the red dots in Figure 6.1. The stress range for the hot spots is 3.3-6.4 GPa.

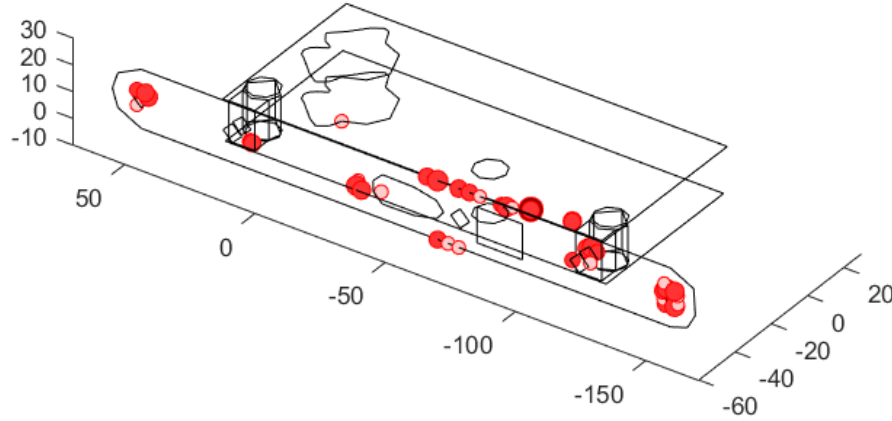


Figure 6.1: *Hot points (red dots) of stresses on lock system. Darker red and larger dots indicates higher stress.*

### 6.2 Convergence studies

From the convergence studies the maximum von Mises stress in each flexible part is analyzed and compared for the different designs. It is also taken into account where the maximum stress occurred, so it is in the same place for all designs.

#### 6.2.1 Time step size

The maximum von Mises stress evaluated in each part for each evaluation design is plotted in Figures 6.2-6.4. The position for the maximum stress on each part is visualized in Figures 6.5-6.7. As seen in the plots the stresses are lower for the largest time step (first design ID, i.e., 20 ms), but by bisect the time step, i.e. 10 ms, a more accurate result is gained.

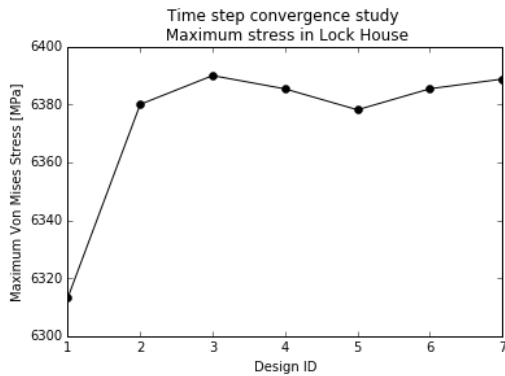


Figure 6.2: *Convergence results of time steps sizes for maximum von Mises stress on the lock house.*

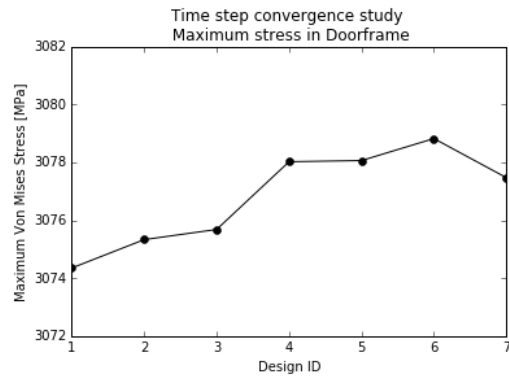


Figure 6.3: *Convergence results of time steps sizes for maximum von Mises stress on the door frame.*



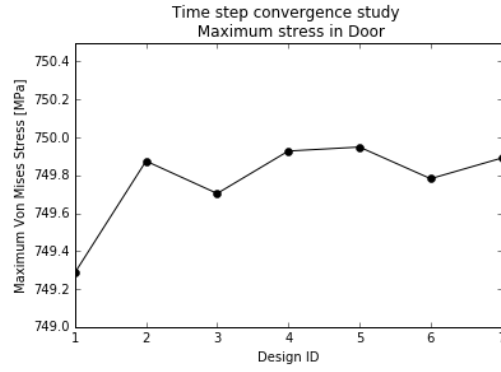


Figure 6.4: Convergence results of time steps sizes for maximum von Mises stress on the door.

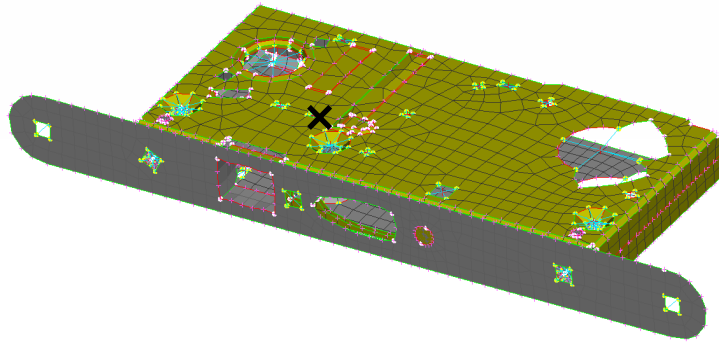


Figure 6.5: The maximum von Mises stress on the lock house is marked with a black cross.

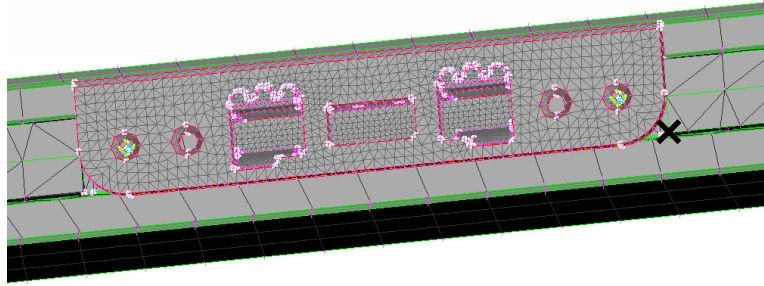


Figure 6.6: The maximum von Mises stress on the door frame is marked with a black cross.

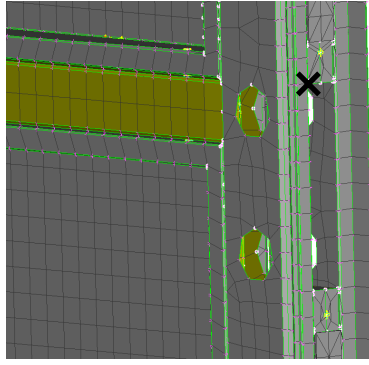


Figure 6.7: *The maximum von Mises stress on the door is marked with a black cross.*

### 6.2.2 Element size

The convergence study of the mesh size gives the von Mises stress in some nodes on the lock house. The results are seen in Figure 6.8. Specified nodes are marked in Figure 6.9.

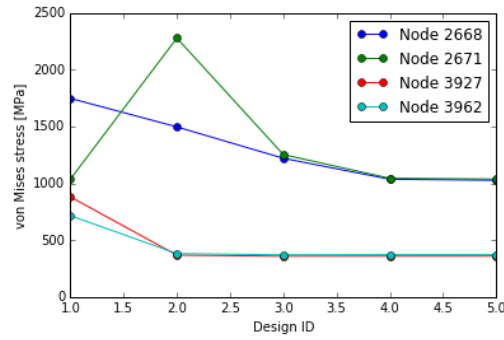


Figure 6.8: *Convergence results of mesh sizes for maximum von Mises stress on the lock house.*

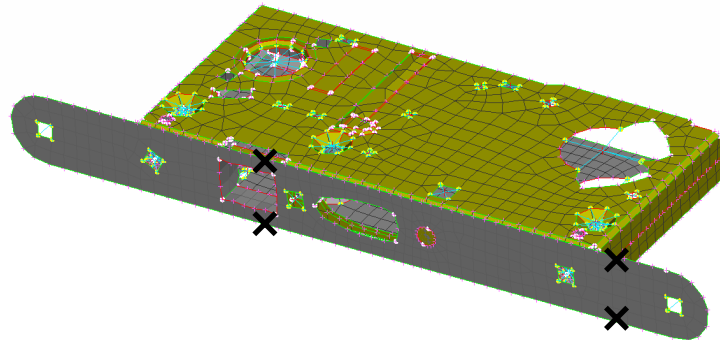


Figure 6.9: *Points (marked with black cross) used to analyze the von Mises stress in the lock house to check for convergence.*

## 6.3 Non linear material model in Adams - MaxFlex

### 6.3.1 MaxFlex model of striking plate

With the force applied on a feature of the striking plate that only is fastened at one of its four sides the total displacement, effective von Mises stress and plastic strain on the striking plate are more or less concentrated to this feature. The maximum displacement is 1.5 mm indicated by red color in Figure 6.10. The maximum effective von Mises stress is approximate 240 MPa and is seen in Figure 6.11. The plastic strain is presented in Figure 6.12 and have a maximum value of 1.7 %.

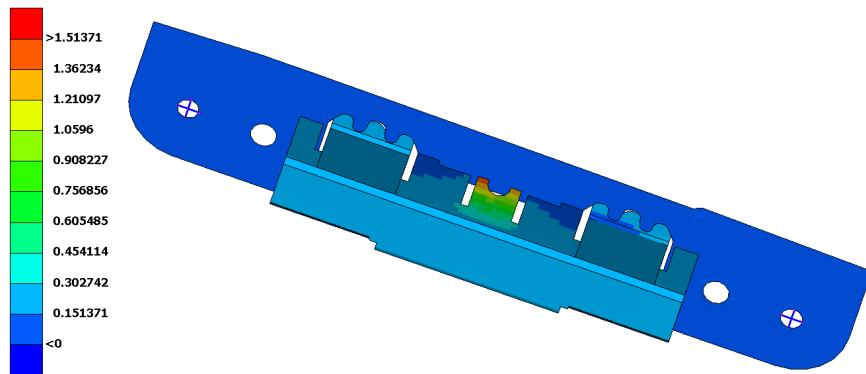


Figure 6.10: *Displacement in mm of the striking plate with a nonlinear material at time 0.6 second with maximum load applied.*

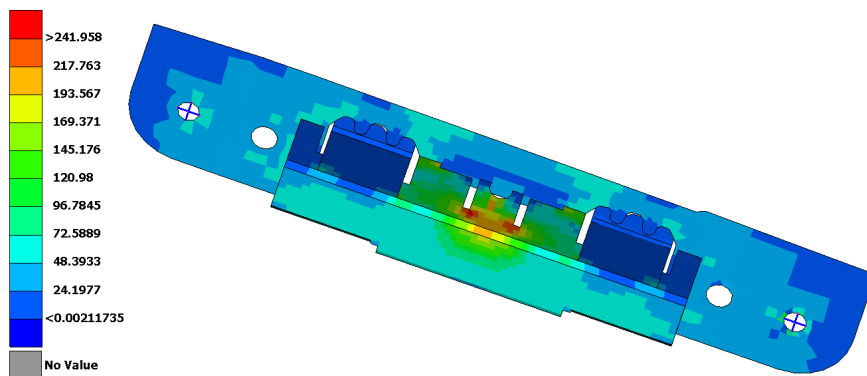


Figure 6.11: *Effective von Mises stress in MPa of the striking plate with a nonlinear material at time 0.6 second with maximum load applied.*

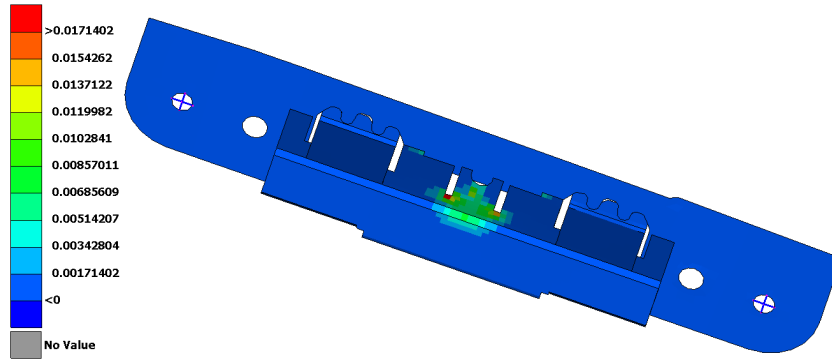


Figure 6.12: *Plastic strain of the striking plate with a nonlinear material at time 0.6 second with maximum load applied.*

### 6.3.2 MaxFlex model of door frame

The simulation of the MaxFlex model of the door frame was allowed to run for 6 days before cancelled due to the long simulation time. During this time the model was simulated 0.489 seconds and reached a maximum force of approximate 11 kN. The displacement, stress and strain result for the MaxFlex model of the door frame are presented in Figures 6.13-6.15. The maximum displacement is 47 mm, maximum stress is 440 MPa and maximum plastic strain is 9 %.

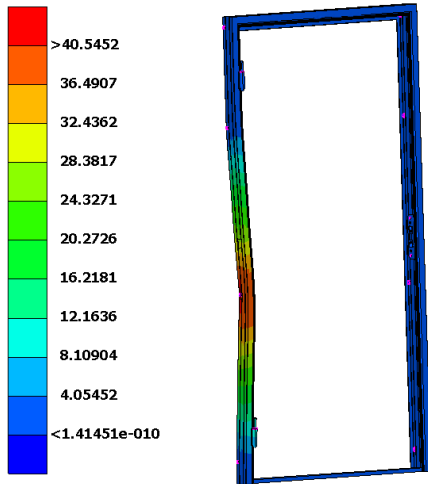


Figure 6.13: *Displacement in mm of the door frame with a nonlinear material at time 0.489 second with maximum load of 11 kN applied.*

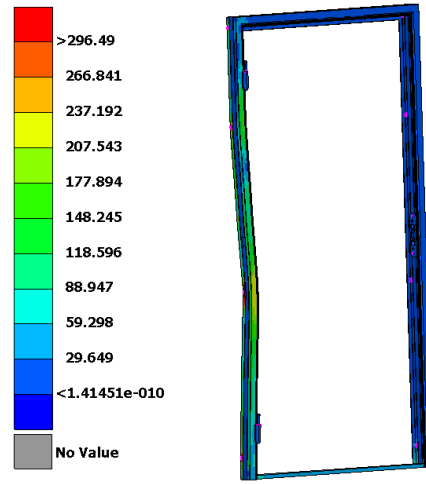


Figure 6.14: *Effective von Mises stress in MPa of the door frame with a nonlinear material at time 0.489 second with maximum load of 11 kN applied.*

The simulation time could have been shortened with other simulation settings and some development of the contacts in the model. Simulation settings that had been a better choice is if the integration tolerance is set to  $10^{-2}$  and the Jacobian pattern is set to *True* (meaning that the Jacobian is evaluated each time). As seen in Figure 6.15 the contacts there need some improvement though the steel now penetrates both itself and the tree, which also slows the evaluation. However, this model will not be further analyzed due to a FE software can handles this problem much more efficient.

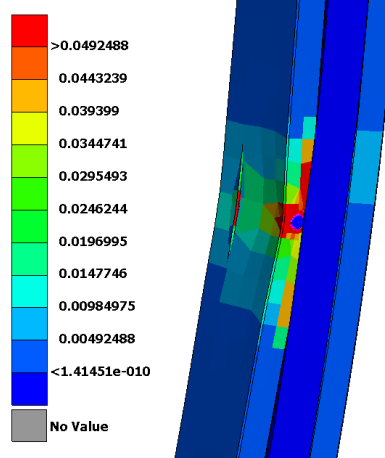


Figure 6.15: *Plastic strain of the door frame with a nonlinear material at time 0.489 second with maximum load of 11 kN applied.*

## 6.4 Verification of Abaqus model of the lock system

The results from the verification of the Abaqus model of the lock system are presented here. The shape of the simulation is similar to the physical test, compare Figure 6.16 with Figure 6.17. Figure 6.18 shows the resulting effective von Mises stress in the model. These results are for evaluation one and give a maximum deformation of approximate 9 mm and maximum stress of 4.9 GPa. The deformation result versus time for each evaluation is shown in Figure 6.19 together with the measure from the physical test. This deformation is measured in the point marked in Figure 6.18. Table 5.8 is presented again for easier understanding of the results, now as Table 6.1.

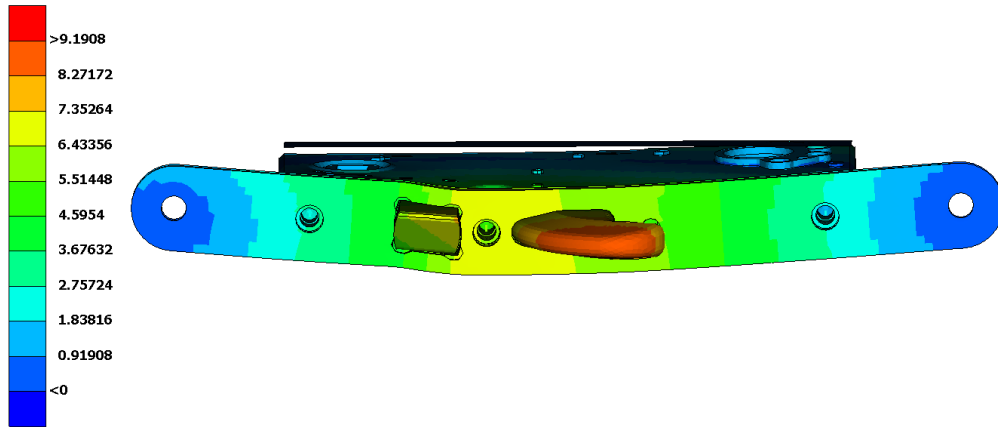


Figure 6.16: *Deformation in mm of the lock system after simulation (evaluation number one).*



Figure 6.17: *Deformation of the lock system after physical test.*

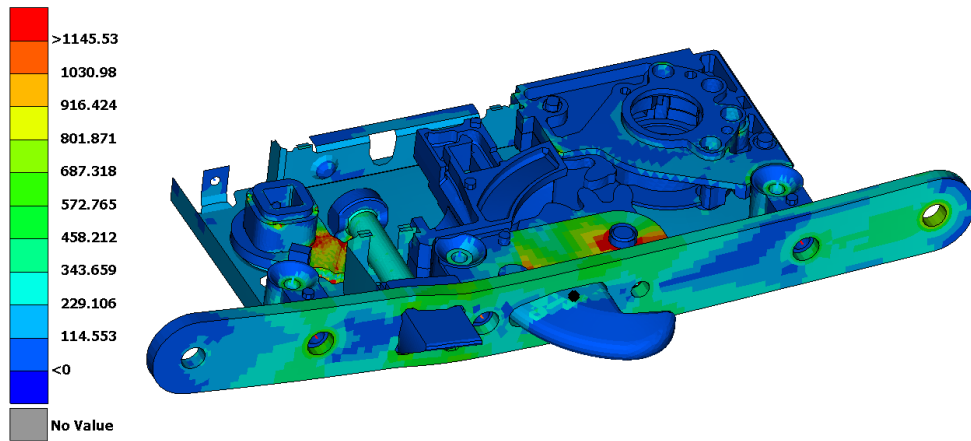


Figure 6.18: Von Mises stress results in MPa from evaluation one. Black cross marked on the Hookbolt is where the deformation is measured.

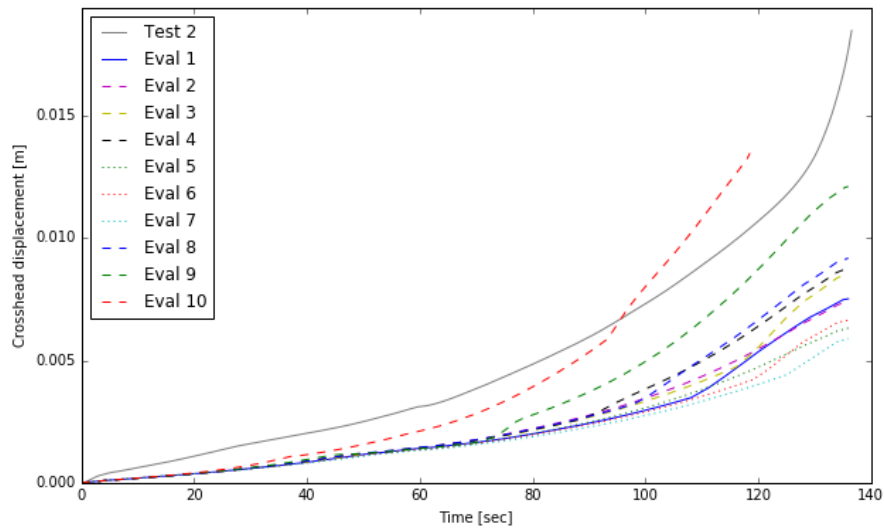


Figure 6.19: Deformation on the Hookbolt over time for each evaluation compared to the result from the physical test, i.e., Test 2 in the legend.

Table 6.1: Translating stress status in MPa for altered material data in each evaluation.

Evaluation	1	2	3	4	5	6	7	8	9	10
Face plate material	-	-30	-	-30	+30	-	+30	-62	-92	-154
EN 1.0327	-	-	-20	-20	-	+20	+20	-37	-55	-93

## 7 Discussion

### 7.1 Software

The model was in the beginning meant to include more mechanisms and movements and therefore Adams was chosen as simulation software. When the project now is almost done, it is clear it became a FE project instead of a project including MBD. If this knowledge had been present in the beginning of the project another software might have been chosen, but the Adams model is still accurate and can be useful for simulations in the future.

The MaxFlex tool in Adams is developed mainly for problems “where the nonlinear behavior of some parts and the motions and loads of the rest of the multibody dynamics (MBD) model influence each other, making accurate results impossible or impractical through separate MBD and FEA analyses” [32]. The tool works for problems only including FEA as well, but though the software is mainly created for mechanisms and movements it requires a small step size which results in unnecessary long simulation times. Most FE software have a more efficient algorithm and can handle larger time steps. Therefore it is easier to shorten the simulation time.

### 7.2 Verification of Abaqus model

The deformation results from the simulation of the Abaqus model of the lock system have a similar shape as the result from the physical test, but with a lower value. This could depend on multiple things. The most likely reason to this is probably the difference of the contacts in the model compared to how it is in reality. In reality the parts are clamped together by the Case and Coverplate with screws, but there will always be a gap between the different parts. In the Abaqus model, each of the parts are tied to the Case, Coverplate or another part and this results in a stiffer model. From the physical test no large movements of the parts could be seen and therefore this assumption is made. However, some unseen movements could exist within the lock house and as the Abaqus model do not allow this movements, the deformation on the model is smaller than reality.

One geometrical difference between the Abaqus model and the lock system used in the physical test is that the Latchbolt is flipped. The simulation (see Figure 6.16) has the Latchbolt’s flat side upwards, while the physical test (see Figure 6.17) has it downwards. The part of the Latchbolt that is hidden inside the lock house is circular and therefore similar in all directions and hence does not impact the result. But where the Latchbolt sticks out from the Face plate, there is one flat side and one slightly rounded side. The rounded side might make it easier for the Face plate to deform, though it both has the opportunity to press straight down in Figure 6.17, but also slide outwards on the Latchbolt. This could mean that the model is less stiff and more deformation occur. The simulation model on the other hand have the flat side upwards towards the pressing Face plate, which could make it harder for the Face plate to deform.

The material data used in the model is not assured correct. It is chosen from a material data base and should correspond approximately. But with some small differs the result changes. Some material modifications are tried out in the different evaluations done, see Figure 6.19. To get approximate the same deformation results as in the physical test, by only translating the material data, the stress need to be translated approximate 40 % of the yield limit for both the nonlinear materials in their negative direction. But as seen in Figure 6.19 even translating the material 50 % (Evaluation 10) does not give the same deformation in the beginning. This indicates that to get same results as in the physical test, with only changing the nonlinear material data, a changed gradient is needed.

## 8 Future work

The Adams model without plasticity (without the use of MaxFlex) can in the future be used for analyzing the stress in each component. It is also possible to analyze the affected result depending on where and how the force is applied and also how different design changes would affect the result. The changes, such as re-dimension or moving any part, can be set as design variables in HEEDS and from there the result for different designs can be analyzed. However, if such an analysis is of interest, a verification of the model might be relevant. There are a multiple areas that could develop the model to mimic the reality better. For example, the screws could be modelled more properly and the parts in the model that currently are merged together may give more realistic result if they were separated. With merged part a too stiff model is gained but by modelling them one by one and applying a contact that allows a small gap between them would make it more realistic. The deformation result could also be stiffer than reality due to that some parts still are rigid bodies and hence it might be good to remake them as flexible bodies.

As the problem is currently defined the MaxFlex track in Adams seems unnecessary. But it could, in the future, be of interest to model the mechanism in the lock system as well. In such a case MaxFlex would be a good choice. It requires some simulation time, but the movements combined with plasticization are difficult to capture in any other simulation programs.

The Abaqus model of the lock system has been compared to a physical test for one loading case, one that is quite similar to the loading case it is exposed for in a break-in. Even though the deformation behave in a similar way, the deformation is not the same. This means that the model needs some further improvement before it is trustworthy for more complex simulations. The first approach could be to change the gradient of the nonlinear material data to see how big of an effect that has on the result. When the model is verified it could be used together with a model of a door to mimic a physical break-in test. To do so a door model needs to be developed and verified and how and where the force should be applied needs to be thoroughly elaborated. This could, however, lead to that fewer physical tests need to be done and a lot of money could be saved.



## References

- [1] *Welcome to the ASSA ABLOY Group - The global leader in door opening solutions*. 2017. URL: <https://www.assaabloy.com/en/com/> (visited on 05/08/2017).
- [2] Swedish Standards Institute. *Svensk Standard SS-EN 1630:2011+A1:2016*. Jan. 19, 2016.
- [3] *BETA CAE Systems - ANSA pre-processor*. 2016. URL: <https://www.beta-cae.com/ansa.htm> (visited on 02/07/2017).
- [4] *Adams*. 2016. URL: <http://www.mscsoftware.com/product/adams> (visited on 01/26/2017).
- [5] *MSC Nastran - Multidisciplinary Structural Analysis*. 2016. URL: <http://www.mscsoftware.com/product/msc-nastran> (visited on 01/26/2017).
- [6] *Abaqus*. 2017. URL: <https://www.3ds.com/products-services/simulia/products/abacus/> (visited on 05/08/2017).
- [7] *HEEDS — Red Cedar Technology*. Apr. 20, 2010. URL: [http://www.redcedartech.com/products/heeds\\_mdo](http://www.redcedartech.com/products/heeds_mdo) (visited on 01/26/2017).
- [8] *CAE Value - Optimization Driven Innovation*. 2017. URL: <http://www.caevalue.com/> (visited on 05/08/2017).
- [9] Japan Automobile Standards Internationalization Center. *FMVSS 214 Side Impact Protection*. URL: <http://125.32.98.1/critenon/%E5%9B%BD%E5%A4%96%E6%A0%87%E5%87%86/FMVSS/FM214E.pdf> (visited on 02/08/2017).
- [10] Red Cedar Technology. *Optimization of a Vehicle Door to Minimize Mass, Rev 12.09*. URL: [http://www.redcedartech.com/pdfs/AB2032\\_DoorMassReductionFinal.pdf](http://www.redcedartech.com/pdfs/AB2032_DoorMassReductionFinal.pdf) (visited on 02/07/2017).
- [11] D. Eby et al. “Shape Optimization of Crashworthy Structures”. Detroit, 2002. URL: <http://www.feainformation.com/m-featured/feb.pdf>.
- [12] *Getting Started with HEEDS v 5.1*. 2007. URL: <http://www.egr.msu.edu/classes/me475/averillr/Lab1/HEEDS.pdf> (visited on 02/07/2017).
- [13] J. E. Bischoff et al. “Advanced Material Modeling in a Virtual Biomechanical Knee”. Newport, Rhode Island, 2008. URL: [http://www.simulia.com/download/solutions/life\\_sciences\\_cust\\_references/BISCHOFF-AUC2008.pdf](http://www.simulia.com/download/solutions/life_sciences_cust_references/BISCHOFF-AUC2008.pdf).
- [14] *HEEDS Exclusive Search Technologies — Red Cedar Technology*. Feb. 21, 2011. URL: [http://www.redcedartech.com/products/heeds/heeds\\_exclusive\\_search\\_technologies](http://www.redcedartech.com/products/heeds/heeds_exclusive_search_technologies) (visited on 01/26/2017).
- [15] N. Chase et al. *A Benchmark Study of Optimization Search Algorithms, Rev 01.10*. 2016. URL: [http://www.redcedartech.com/pdfs/SHERPA\\_Benchmark\\_0110.pdf](http://www.redcedartech.com/pdfs/SHERPA_Benchmark_0110.pdf) (visited on 02/07/2017).
- [16] *ASCII*. Wikipedia. Page Version ID: 761914408. Jan. 25, 2017. URL: <https://en.wikipedia.org/w/index.php?title=ASCII&oldid=761914408> (visited on 01/26/2017).
- [17] C. E. Mackenzie. *Coded Character Sets, History and Development*. United States of America: Addison-Wesley Publishing Company, Inc., 1980. ISBN: 0-201-14460-3. (Visited on 01/27/2017).
- [18] *Ancient Computer Character Code Tables – and Why They’re Still Relevant*. 2016. URL: <http://blog.smartbear.com/development/ancient-computer-character-code-tables-and-why-theyre-still-relevant/> (visited on 01/26/2017).
- [19] *Adams MaxFlex*. 2016. URL: <http://www.mscsoftware.com/product/adams-maxflex> (visited on 04/10/2017).
- [20] *Eastern Alloys - Zinc Die casting Alloy (ZAMAK 2, ZAMAK 3, ZAMAK 5, ZAMAK 7, ZAMAC, ZA-8, ZA-12, & ZA-27), Zinc Ingots*. 2016. URL: <http://www.eazall.com/zinc-die-casting-alloys> (visited on 01/27/2017).
- [21] *16NiCrS4*. URL: <https://steelnavigator.ovako.com/steel-grades/16nicrs4/> (visited on 03/06/2017).
- [22] *Technical card C45E*. 2015. URL: [http://www.lucefin.com/wp-content/files\\_mf/10c45e70.pdf](http://www.lucefin.com/wp-content/files_mf/10c45e70.pdf) (visited on 01/27/2017).
- [23] *AISI 1010 Steel, cold drawn*. 2016. URL: <http://www.matweb.com/search/datasheet.aspx?matguid=025d4a04c2c640c9b0eaaef28318d761> (visited on 01/26/2017).
- [24] *S275JR steel plate, S275JR sheet, S275JR coil - Carbon steel -*. URL: <http://www.steelss.com/Carbon-steel/s275jr.html> (visited on 03/06/2017).
- [25] *Specifications for Common Plastic Molding Design Material — Engineers Edge*. URL: [http://www.engineersedge.com/plastic/materials\\_common\\_plastic.htm](http://www.engineersedge.com/plastic/materials_common_plastic.htm) (visited on 01/27/2017).
- [26] *S235JR*. Jan. 31, 2017. URL: <https://steelnavigator.ovako.com/steel-grades/s235/?variantIDs=702> (visited on 03/06/2017).

- [27] *Träfiberskivor - TräGuiden*. URL: <http://www.traguiden.se/om-tra/material-et-tra/trabaserade-produkter/skivmaterial1/trafiberskivor/> (visited on 01/30/2017).
- [28] *Medium Density Fiberboard (MDF) :: MakeItFrom.com*. 2016. URL: <http://www.makeitfrom.com/material-properties/Medium-Density-Fiberboard-MDF> (visited on 01/30/2017).
- [29] *Glas som byggmaterial*. 2013. URL: <http://www.pilkington.com/europe/sweden/swedish/building+products/glasfakta/grundläggande+om+glas/glas+som+byggmaterial/default.htm> (visited on 01/31/2017).
- [30] *RISE - Research Institutes of Sweden*. 2016. URL: <http://www.sp.se/en/Sidor/default.aspx> (visited on 01/26/2017).
- [31] *Total Materia - The world's most comprehensive materials database*. 2016. URL: <http://www.totalmateria.com/page.aspx?ID=Home&LN=EN> (visited on 05/09/2017).
- [32] *Adams MaxFlex, Complete Nonlinear Flexibility in Multibody Dynamics Simulations, SOLUTION BRIEF*. 2015. URL: [http://media.mscsoftware.com/sites/default/files/sb\\_adams-maxflex\\_ltr\\_w.pdf](http://media.mscsoftware.com/sites/default/files/sb_adams-maxflex_ltr_w.pdf) (visited on 05/28/2017).

# A BDF for MNF generation of the flexible bodies

```

$$$$-----$
$$              Executive Control Cards              $
$$$$-----$
INIT MASTER(S)
NASTRAN BUFFSIZE=65537
NASTRAN SYSTEM (151)=1
NASTRAN SYSTEM(107)=4
GEOMCHECK NONE
SOL 103
CEND
ADAMSMNF FLEXBODY=YES,FLEXONLY=YES,ADMOUT=NO,OUTGSTRN=NO,OUTGSTRS=YES
$$$$-----$
$$              Case Control Cards                  $
$$$$-----$
ECHO=none
METHOD=2
STRESS(PLOT)=ALL
GPSTRESS(PLOT)=ALL
OUTPUT(POST)
SET 1 = ALL
VOLUME 1 SET 1
SET 2 = ALL
SURFACE 2 SET 2
$$$$-----$
$$          SET DEFINITION SECTION and BULK DATA card in include file          $
$$$$-----$
include (INPUT FILENAME OF NASTRAN-FILE)
$
DTI,UNITS,1,MGG,N,MM,SEC
$
PARAM,AUTOQSET,YES
$
$ Allow RBE3 nodes in ASET
$PARAM,AUTOMSET,YES
PARAM,AUTOSPC,YES
$ Overview of masses, CoG, etc.
PARAM,GRDPNT,0
$ Do not print grid point singularity table
PARAM,PRGPST,NO
$
$$$$-----$
$              EIGRL cards                        $
$$$$-----$
EIGRL          2              30          4
$
$
ENDDATA

```

## B Simulation script used in HEEDS for running the Adams simulation

```
file/model=ASSA_000054084.adm, output=simulation  
output/noseparator  
simulate/transient, end=4, steps=500  
stop
```

## C BDF parameters for the MaxFlex model of the striking plate

```

$$$$-----$
$$                               Executive Control Cards                               $
$$$$-----$
INIT MASTER(S)
NASTRAN BUFFSIZE=65537
NASTRAN SYSTEM (151)=1
NASTRAN SYSTEM(107)=4
SOL 103
CEND
ADAMSMNF FLEXBODY=YES,FLEXONLY=YES,ADMOUT=NO,OUTGSTRN=NO,OUTGSTRS=YES
$$$$-----$
$$                               Case Control Cards                               $
$$$$-----$
ECHO=none
METHOD=2
STRESS(PLOT)=ALL
GPSTRESS(PLOT)=ALL
OUTPUT(POST)
SET 1 = ALL
VOLUME 1 SET 1
SET 2 = ALL
SURFACE 2 SET 2
$$$$-----$
$$          SET DEFINITION SECTION and BULK DATA card in include file          $
$$$$-----$
$
$ANSA_VERSION;17.0.3;
$
(INsert NAS-OUTPUT OF MODEL)
$
DTI,UNITS,1,MGG,N,MM,SEC
$
PARAM,AUTOQSET,YES
$
$ Allow RBE3 nodes in ASET
PARAM,AUTOSPC,YES
$ Overview of masses, CoG, etc.
PARAM,GRDPNT,0
$ Do not print grid point singularity table
PARAM,PRGPST,NO
$
$-----$
$                               EIGRL cards                               $
$$$$-----$
EIGRL          2                      30          4
$
ENDDATA

```

## D BDF parameters for the MaxFlex model of the door frame

```

$$-----$
$$                               Executive Control Cards                               $
$$-----$
INIT MASTER(S)
NASTRAN BUFFSIZE=65537
NASTRAN SYSTEM (151)=1
NASTRAN SYSTEM(107)=4
SOL 103
CEND
ADAMSMNF FLEXBODY=YES,FLEXONLY=YES,ADMOUT=NO,OUTGSTRN=NO,OUTGSTRS=YES
$$-----$
$$                               Case Control Cards                               $
$$-----$
ECHO=none
METHOD=2
STRESS(PLOT)=1
GPSTRESS(PLOT)=ALL
OUTPUT(POST)
SET 2 = ALL
SURFACE 2 SET 2
$$-----$
$$          SET DEFINITION SECTION and BULK DATA card in include file          $
$$-----$
$
$ANSA_VERSION;17.0.3;
$
(INsert NAS-OUTPUT OF MODEL)
$
DTI,UNITS,1,MGG,N,MM,SEC
$
ASET1,200000,200101,200102,200201,200202,200203,200204,200205,200206,200207,200208
$
PARAM,AUTOQSET,YES
$
$ Allow RBE3 nodes in ASET
$PARAM,AUTOMSET,YES
PARAM,AUTOSPC,YES
$ Overview of masses, CoG, etc.
PARAM,GRDPNT,0
$ Do not print grid point singularity table
PARAM,PRGPST,NO
$
$-----$
$                               EIGRL cards                               $
$$-----$
EIGRL          2                      30          4
$
ENDDATA

```