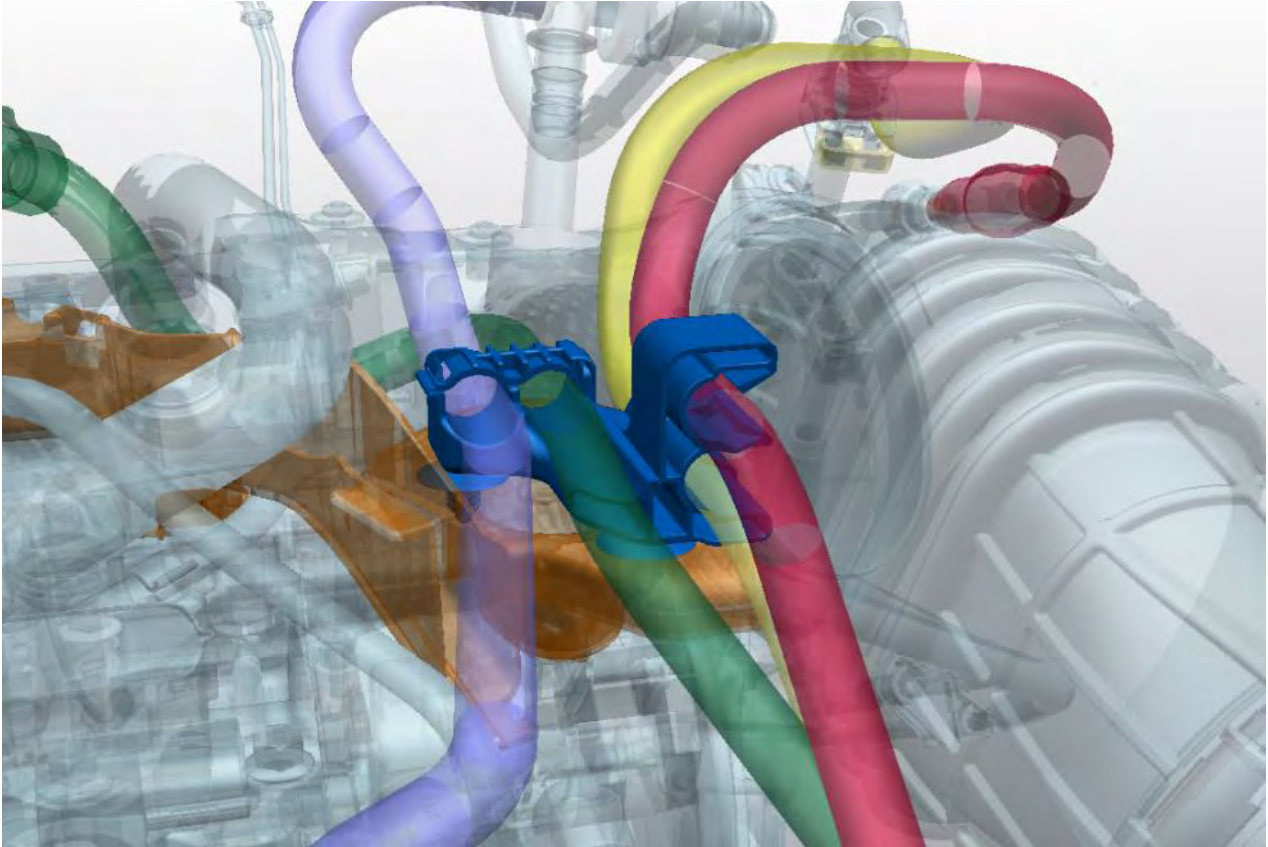




CHALMERS
UNIVERSITY OF TECHNOLOGY



Clips and Brackets. Assembly & Disassembly Force

Master's thesis in Applied Mechanics

Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt

MASTER'S THESIS IN APPLIED MECHANICS

Clips and Brackets. Assembly & Disassembly Force

Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt

Department of Applied Mechanics
Division of Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2017

Clips and Brackets. Assembly & Disassembly Force
Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt

© Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt, 2017-06-01

Master's Thesis 2017: 16
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:
New clip design model of Volvo Car Corporation (Blue) holding the four tubes, and
the fixation base (Orange); Obtained from Teamcenter software.

Name of the printers / Department of Applied Mechanics
Göteborg, Sweden 2017-06-01

*A nuestras familias,
gracias por todo*

Clips and Brackets. Assembly & Disassembly Force
Master's thesis in Applied Mechanics
Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt
Department of Applied Mechanics
Division of Dynamics
Chalmers University of Technology

Abstract

Some of the core values shared by Volvo Car Corporation are safety and health of their employees, for this reason the company needs fast, reliable and accuracy tools that allows the verification of security and ergonomic criteria's. This research is focused on simulating the forces needed to assembly and disassembly the fuel lines into the clip all around the fuel system, since the actual verification method is the real test made after its manufacture. In order to obtain a good simulation tool, during the research period several task are performed such us: simulations of two different clip models with their respective parameter changing and setting, real tests of those models, study of the behaviour of the material, correlation between results and redesigns in one of the models. The obtained results and the correlations show different conducts, in some cases, the simulation gives values really close to the reality, and in other cases, the values are a bit different. However, in general terms the behaviour of the clip is really close to the reality. As a result, it can be assumed that the simulation's methodology achieved could not replace the actual verification method, but it can be used as a verification method for the early stages, in a complementary way, allowing monetary and time saving.

Key words: Fuel lines; Automotive; Simulation; FEM; Abaqus; ANSA; Test; Polyamide; Volvo Car Corporation; Assembly force; Pull-off force; Correlation; Recommendations; Design

Contents

Abstract	I
Contents	III
Preface.....	V
Notations	VI
1 Introduction.....	1
1.1 Background	1
1.2 Gap	2
1.3 Objective & scope	2
1.4 Methodology	2
1.4.1 Simulation.....	3
1.4.2 Testing and specifications.....	3
1.4.3 Correlation	3
1.4.4 Methodology flowchart.....	4
1.4.5 Report outline.....	4
2 Finite element method.....	5
2.1 The basis of the method	5
2.2 FEM in abaqus	5
2.2.1 Elements.....	5
2.2.2 Linearity, nonlinearity and analysis type	8
3 Models of study.....	9
3.1 Clip classification.....	9
3.2 Requirements.....	10
3.3 Models.....	10
3.3.1 Specifications.....	11
3.3.2 Material	12
4 Clip simulation.....	14
4.1 Simulation overview	14
4.1.1 Pre-processing.....	14
4.1.2 Computer solving.....	15
4.1.3 Post-processing	15
4.2 Abaqus structure.....	16
4.3 Simulation iterations	17
4.3.1 Pre-study	17
4.3.2 Models development.....	18
4.3.3 Results.....	42

4.3.4	Redesign.....	45
5	Real tests	47
5.1	Environmental conditions	47
5.2	Pre-test.....	47
5.3	Equipment	49
5.4	Testing steps and procedure	50
5.5	Post- test	53
5.6	Test results.....	53
6	Results correlation	57
6.1	Considerations	57
6.2	Old design model	58
6.3	New design model.....	59
7	Recommendations for upcoming designs	65
7.1	Symmetry	65
7.2	Thickness of the wings.....	66
7.3	Corners, edges & shapes	66
8	Conclusions.....	68
9	Technical recommendations for clip simulation.....	70
9.1	Settings	71
9.2	Import geometry	71
9.3	Check & clean up	72
9.4	Defeaturing.....	73
9.5	Creating tubes.....	74
9.6	Mesh	77
9.7	Properties and materials	81
9.8	Sets	82
9.9	Boundary conditions	83
9.10	Contacts	84
9.11	Step	86
9.12	Export	88
9.13	Reading output files.....	88
9.14	Visualization.....	89
10	Future work.....	94
11	References.....	95
	Appendix.....	96
	Appendix 1, Contribution report.....	97
	Appendix 2, Full results real tests.....	98

Preface

The following project “Clips and Brackets. Assembly & Disassembly Force” has been carried out at Volvo Cars Corporation, in the Fuel System department within the Research & Development organization, in Göteborg, Sweden, during the Spring term of 2017. This study is presented as Master Thesis Work for Chalmers University of Technology at the Applied Mechanics department.

The performance of the project has been carried out thanks to the supervision of Parvin Zarabadi, Roar Matsson and Kjell Hävell at Volvo Cars Corporation and Thomas Abrahamsson as the supervisor/examiner at Chalmers University of Technology.

We would like to stress the support and help received from Johanna Lilja at the CAE Durability Body & Trim department and Adriano Linero Jiménez at the Material Centre department. Without their respective wide knowledge in finite element analysis and experience in real testing, the project would not have been possible.

Finally, we would also like to thank the engineers of the Department of Fuel System and EVAP to be involved in this Thesis. Also thanks the CAE and Material Centre departments and all the co-workers for their great reception.

Göteborg June 2017-06-01

Mariana Fernández del Olmo Heres
Alejandro Fernández Schmidt

Notations

Explained below are the variables that the reader will find within the report, with its respective explanation and meaning.

a	Acceleration
BC	Boundary condition
CAD	Computer aided design
CAE	Computer aided engineering
$CONS$	Curves on surface
d	Displacement
DB	Database
DOF	Degree of freedom
f	Output frequency
FEM	Finite element method
I	Internal force
m	Mass
N	Newton
p	Contact pressure between surfaces in contact
P	External force
RF	Reaction force
RH	Relative humidity
RT	Room temperature
s	Second
t	Time
TR	Technical requirement
v	Velocity
$3D$	Three-dimensional
X_s	X local coordinate of profile line start point, in an analytical rigid tube
X_e	X local coordinate of profile line end point, in an analytical rigid tube
Y_s	Y local coordinate of profile line start point, in an analytical rigid tube
Y_e	Y local coordinate of profile line end point, in an analytical rigid tube
(Xa, Ya, Za)	Coordinates of the starting point “a” defining the tube axis
(Xb, Yb, Zb)	Coordinates of the ending point “b” defining the tube axis
τ_{crit}	Shear stress in a contact
μ	Friction coefficient
✓	Fulfill
✗	Do not fulfil

1 Introduction

In this section, the reader finds the background of the project, the objective, the project's scope and its working method.

1.1 Background

Nowadays most of the car manufactures devote huge effort and time to safety. Among them Volvo Car Corporation highlights the safety compromise as one of its main values. The vehicle fuel system shall be one of the safest system in the car, regarding the potentially hazardous conditions that can appear with a bad behaviour.

Keeping all components in the fuel system fixed in the right position is fundamental during the minimum 15-year and 240.000 km car life, but at the same time, the assembly forces should be low enough in order to reduce RSI (Repetitive strain injuries) for factory workers.

One of the most difficult components to deal with, are the fuel lines, since they are installed in harsh environments and crash zones, from the front to the rear of the vehicle. Clips retain and keep in a proper place the fuel line, performing their fixation, and must fulfil several technical, ergonomic, safety and legal requirements.

The clip design used to belong entirely to supplier companies specialized in automotive fluid systems, but currently most of the car companies are involved in an internal renewal process, seeking keep the knowledge indoors, trying to save time, and especially money. Volvo Car Corporation retains as some of its fundamental principles the safety and the idea of making people's live easier, so in this context the idea of developing their own clip designs must be linked to a process that firmly ensures these values.

The simulation of different pieces or systems of the vehicle gives accurate and complete results of their real behaviour, that is, from a single model simulation one can obtained different outputs (such us strains, forces, energies, stresses, etc.) in all the different parts of the piece, which is impossible to obtain on a real test. The clear advantages of the simulation against the real testing are related with time and money saving and the depth of the analysis, not only in the automotive industry, but also in all the engineering fields. Inconvenience about it is the previous correlation that needs to be done within real test and simulation, since the simulation tools must represent the real behaviour, and that is a really complex task.

The huge time and money saving is obvious for different reasons but the main one is related with the production. When performing a real test of any piece or system, the real manufactured piece is required, it means that the machine tool that manufacture the piece is also needed. Therefore, in most of the cases, the pieces are tested during or after production. In case that the real testing shows an undesired behaviour, the production of that part should be removed and changed. The huge amount of money and time could be incalculable (depending on the part), and with a correct simulation tool that could be saved. Moreover, but in a discreet way, the testing need a few samples to be done.

1.2 Gap

With this study is intended to deepen the existing research about clip design and seek to create developments that can bring new knowledge and procedures.

There are investigations related to specific clip simulations, but not a simulation method that can be applied to different clip models. Currently, the real test is established as the reliable method of verification for fulfilling the technical requirements. It is noticeable the lack of a generic and accurate simulation method to perform those verifications, or at least to have a general overview of the clip behaviour. It would allow to analyse the clip's requirements in their early stages, when the clip is just a CAD model, without the need of obtaining the manufactured part, which is really important in terms of saving time and money.

With a greater degree of collaboration between the design area and the simulation field, the company could obtain a better behaviour of the clip. In the same way, an improved and systematic method when testing clips would establish a connection for the designers and the real tests. That could also achieve better and more accurate results, at the same time of some hints and tips for the designers.

1.3 Objective & scope

The purpose of the thesis is to increase the knowledge about clip design through testing, simulating and analysing existing clips in order to implement a simulation method and to improve the existing test method that will be used in upcoming new designs.

The study focuses on the assembly and pull-off forces that are necessary to achieve the inclusion of the fuel line into the clip and the holding of the line on it. In order to understand and reduce the ergonomic risks for factory workers and check the fulfilment of the technical requirements, with a fundamental emphasis on safety.

Outside of this project's scope are the assembly and pull-off forces needed to insert and remove the clip from the part where is attached to. Also the pull-off force of clip's holes with locker will not be part of the study. Theoretical calculations are neither include due to its complexity.

Therefore, the study contains the simulations of certain relevant clips, with their respective parameters variation in order to obtain the desired result, possible real testing method improvement and results correlation between this last one and the simulations. Also contains technical recommendations for a suitable simulation and possible design recommendations for new clip designs.

1.4 Methodology

The developing of the project is based on two main tasks implemented simultaneously, the simulation of clips and the performing of real tests. Through out a results correlation between both of them, technical recommendation for a suitable clip simulation and some recommendations for upcoming designs will result as goals of the thesis.

1.4.1 Simulation

The first task is to perform the simulations of the two clip models. The main tools used for the simulation are Teamcenter, CATIA V5, ANSA (with an Abaqus module) and Abaqus viewer. With the purpose of giving the reader an overall knowledge of the simulation process, the procedure is the following: The CAD model is obtained from CATIA V5, a strong computer aided design software.

Teamcenter is a set of PLM (product lifecycle management) software applications that for this study make the searching of any component of the vehicle easier. The CAD file, the temporary evolution of the design, the part's surroundings and more information is available. Therefore, this tool allows to locate the work piece and obtaining all the information that may be needed.

ANSA is an advanced CAE pre-processor with several functions. This software allows read the CAD file and make the clean-up of that model. Different modules on ANSA are available depending on the solver used (Abaqus module for this study). This means that ANSA will create a specific format file depending on the module that one is working with. The file format with ANSA software is *.ansa*.

Once the model is ready, ANSA allows to create an *.inp* file. The file is sent to the general cluster and is queued to perform the finite element analysis on the Abaqus/Standard CAE solver. Once is solved, an *.odb* file is send back to the owner. To conclude the analysis, the Abaqus Viewer software opens that file and transform all the operations and data obtained in a visual format much easier to analyse.

The reader must understand that the finite element method analysis may find some problems during the solving of the implemented problem.

It is remarkable that during the development of this study the solving convergence issues are mentioned several times. In convergence problems the solver does not achieve a solution, by not complying some criteria previously established for the solver, as could be: convergence tolerance parameters for force or moment, time incrementatiton control parameters or contact controls.

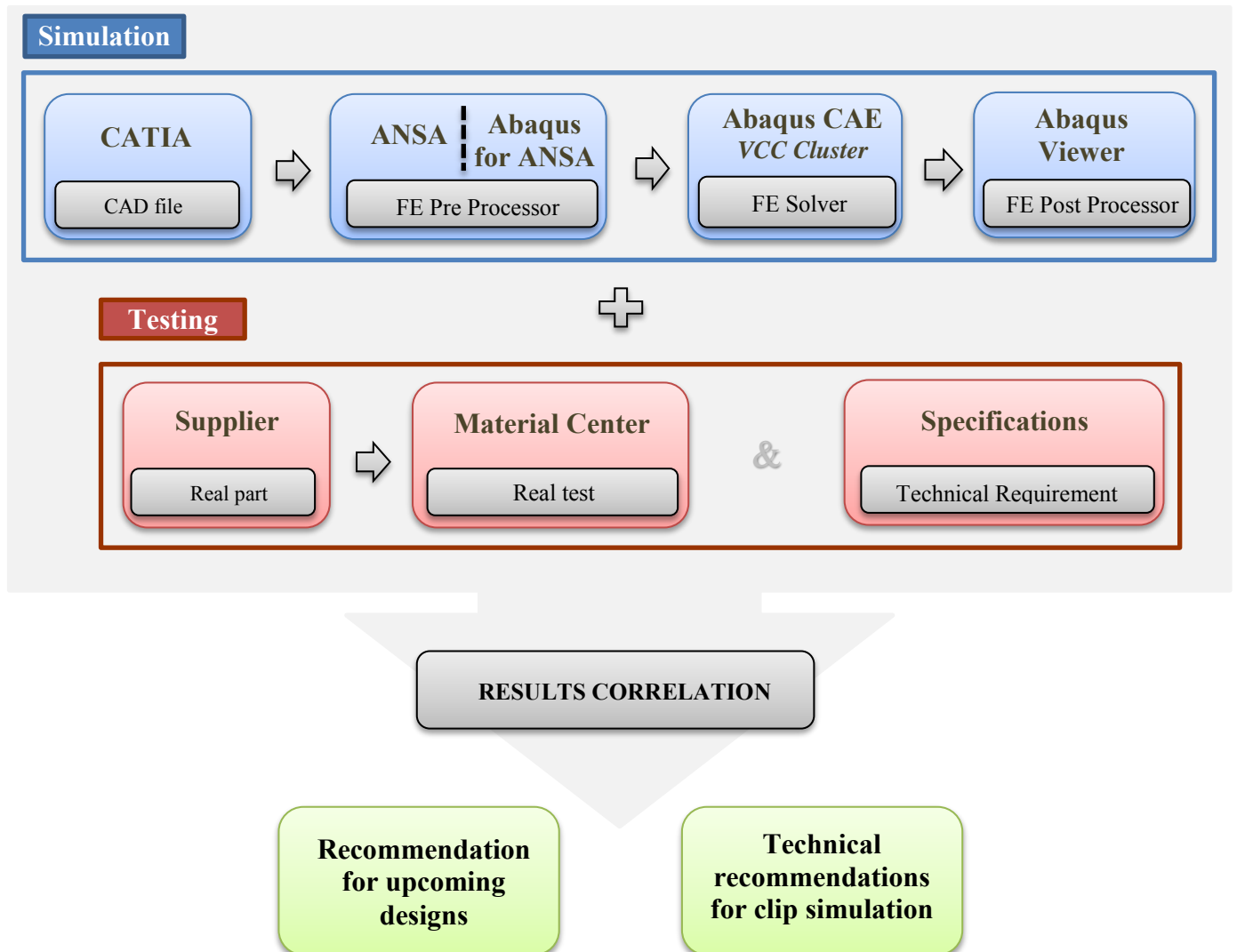
1.4.2 Testing and specifications

The second task is perform the testing, which needs the physical part obtained from the supplier. The testing process is carried out in the Material Centre, with the help of a tensile test machine and the TestExpert II software. Furthermore, the test has to be done based on the Technical Requirement. The whole procedure is explained in detail on the Chapter 5 Real tests.

1.4.3 Correlation

After completing both tasks, a correlation of the results is necessary in order to verify the validity of the simulations. During this study, the correlation is always present, in a iterative way, so that each simulation can be improved. The final correlation includes comparisons between the forces obtained, the strains and the behaviour of the clip, and it will show the level of accuracy of the simulating tool.

1.4.4 Methodology flowchart



1.4.5 Report outline

Within this document the reader will go through different chapters that will allow him to understand the carried out research project. The document's main parts are the followings: finite element method, models to study, clip simulation, real test, results correlation, recommendation for upcoming designs, conclusions, technical recommendations for clip simulation and possible future work.

2 Finite element method

2.1 The basis of the method

The finite element method is a methodology that studies and analyses displacements, deformations and tensions generated on a body that is subjected to certain loads and restrictions. The variations of the method allows also making other analysis such as resistance, rigidity, fatigue, dynamic, acoustic or thermal analysis.

Any geometry or structure, either a 2D or 3D solid, is formed by infinite unknown elements, which obviously make the analysis impossible. What FEM makes is to divide the geometry into a finite number of elements, whose geometry and location are known. The points that join two or more of these elements are called nodes or nodal points, and the general set of elements and nodes that conforms the whole geometry is called mesh. The discretization process that the mesh carries out makes the analysis of the body possible, since the mathematical equations that govern the behaviour of each element can be applied, and extrapolated to the whole body.

The accuracy and quality of the analysis strongly depends on the mesh. In general terms, big size elements and low number of elements (rough mesh) results on a bad simulation that does not represent the real behaviour of the studied part. On the contrary, the reader could think that the smaller the element sizes are and higher the number of elements are (fine mesh), the best the results will be. That is partly true, since it is recommend to use a fine mesh to get as close to reality as possible results, but there are two things that need to be considered. Firstly, there is a certain moment when the mesh tends to converge, that is, the simulation shows the same results from a mesh size and its successive lower sizes. And secondly, a highly fine mesh means performing lots of operations, which could be translated as excess of data handling, and the consequent slowdown of the process.

In all the studies is interesting to make a comparison between the solutions provided by the software and the solutions achieved by making some simplified calculations of the model. Unfortunately, carrying out a simplification of this 3D model so that it can be studied on a paper is extremely complex. The fact of performing a theoretically calculation simplifying the model to obtain a result that would not resemble reality, would mean a high consumption of time and work. The possibilities that the software offers simplify performing a roughly analysis (see Chapter 4, Section 4.3.1 Pre-study).

2.2 FEM in abaqus

2.2.1 Elements

There are different types of elements available in Abaqus. Which normally are characterized by aspects like family, DOF, number of nodes, formulation or integration. Each of them receive a unique name, identifying the five parameters above mentioned.

2.2.1.1 Family

The first letter of an element's name, refers to which family belongs to. (for instance, M3D, means a membrane element)

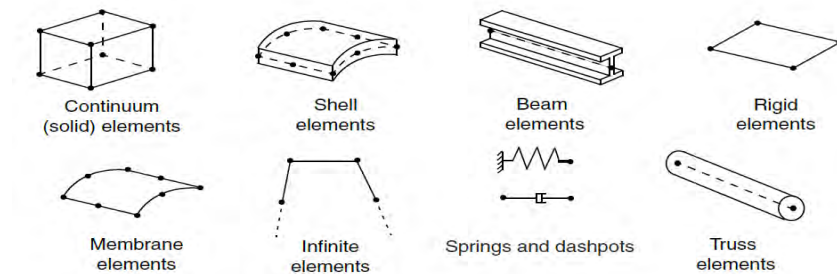


Figure 1. Family of elements in Abaqus; Obtained from Getting started: Abaqus interactive edition.pdf

2.2.1.2 Dimensionality

Usually the two letters following the “family character” in the element's name, refer to the extension in a certain direction as a property of space:

- Character **3D**: three-dimensional element.
- Character **AX**: axisymmetric element.
- Character **PE**: Plane strain element.
- Character **PS**: Plane stress element.

2.2.1.3 Number of nodes – order of interpolation

One should notice that in FEM, the displacements, rotations, etc. are calculated only at the nodes of the element. Moreover, at any other point of the element, the desired results are obtained by interpolation from the nodal results.

Here is found a key point in the development of a model, depending on where the nodes are situated it is possible to distinguish between:

- Linear elements/1st order elements: Which have nodes only in their corners, obtain the results using a linear interpolation.
- Quadratic elements/2nd order elements: Which have nodes in the middle of each edge, obtain the results using quadratic interpolation.

- Modified 2nd order elements: Triangular or tetrahedral elements with mid-edge nodes, obtain the results using modified 2nd order interpolation.

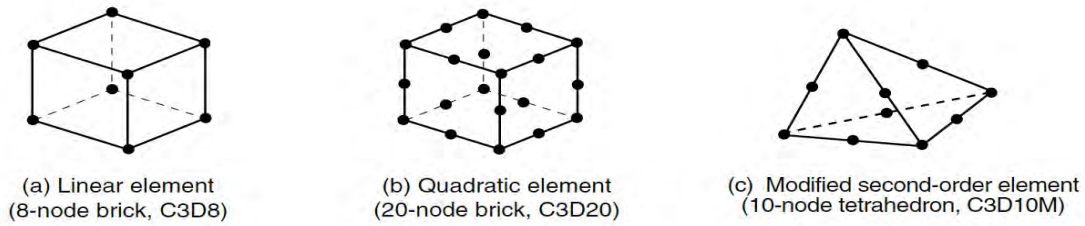


Figure 2. Type of elements (DOF) in Abaqus; Obtained from *Getting started: Abaqus interactive edition.pdf*

The number that normally follows the letter (family) and dimensionality, refers to the number of nodes in the element or the order of interpolation that the element uses, depending on the family that belongs to:

- Continuum or Shell family: for instance C3D8, brick of eight nodes in the corners.
- Beam or Membrane family: for instance B31, 3D beam with 1st order interpolation.

2.2.1.4 Formulation

The formulation refers to the mathematical theory that govern the element's behaviour. There are two different types of formulations for the behaviour of an element, depending on if the material can *-Eulerian-* or cannot *-Lagrangian-* flow across the element boundaries.

Eulerian description is used in fluid mechanics simulations, so it will not be used in this study case and *Lagrangian* formulation is chosen.

Some elements in ABAQUS have some alternative formulations, which can be identified by the last letter of the element's name:

- Character **H**: Hybrid formulation, which for instance includes the treatment of the pressure in Continuum elements or axial force in Beam elements.
- Character **T**: Thermal formulation, including the thermal DOF.
- Character **I**: Incompatible mode formulation.

2.2.1.5 Integration

Abaqus uses numerical techniques to integrate quantities over the volume of an element, being the Gaussian quadrature the most common. Some elements in Abaqus can be integrated in a fully or reduced manner, so that one can find a letter **R** on the end of the element's name, referring to the reduced-integration version.

2.2.2 Linearity, nonlinearity and analysis type

A linear problem has a relationship between the applied loads and the response on the system. But contrary in a nonlinear problem model's stiffness is changing as it deforms.

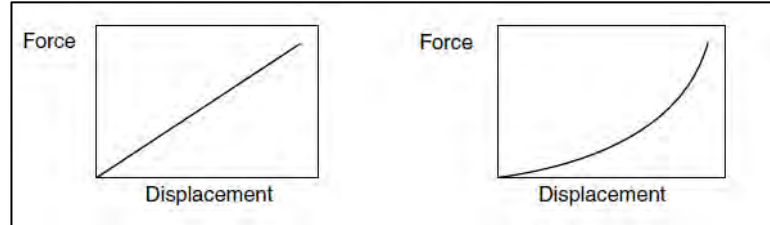


Figure 3. Linear and non-linear curves; Obtained from: *Getting started: Abaqus interactive edition.pdf*

Linear analysis is valid for simple purposes, but has as limitation the difficulty of achieve convergence in the solution. Therefore and due to the complexity of some model's parameters, as contacts and frictions, in this study a nonlinear problem is established. The main sources of nonlinearity in the studied models are material nonlinearity, which is related with the changes in material properties, and geometric nonlinearity due to the effect of the large displacements in the model.

The analysis type is related with the model's "linearity". For a nonlinear dynamic analysis two different Abaqus products can be used. The first one is Abaqus/Standard, which uses Newton-Raphson method, combining increments and iterations to obtain the solution. Considering external forces (P) and internal forces (I) acting on the model, the aim of this solver is to achieve static equilibrium:

$$P - I = 0 \quad (2.1)$$

Abaqus/Standard splits the running simulation in load increments trying to find the equilibrium in each iteration. Usually it takes some increments to achieve a valid solution and the sum of all responses is the approximate solution for the nonlinear analysis.

The second one is Abaqus/Explicit that proceeds seeking the solution for dynamic equilibrium equation (see equation (2.2)) without iterating by explicitly advancing the kinematic state from the end of the previous increment. That could avoid convergence problems in some models but add a risk factor in the solution as explained below (Getting started with Abaqus: Interactive Edition, 2012).

$$P - I = ma \quad (2.2)$$

Although Abaqus/standard requires a longer running time to obtain the solution due to the high number of solved equation in all of the increments and iteration, but despite this, remains the chosen option. The main reason is the risk of obtaining undesired results with Abaqus/Explicit caused by a directly and not iteratively way of obtain the solution. An Abaqus/Explicit analysis could be used in future simulations having previously gained experience with current models (Getting started with Abaqus: Interactive Edition, 2012, pp 247).

3 Models of study

In this chapter, basic information, requirements and characteristics of the clips for this study are introduced, in order to situate the reader in the research knowledge field.

Clips for fuel lines are retention devices that shall keep the line retained to structural components or retained between tubes and pipes (distance clips). The clip design is made taking into account the available space in terms of packing, so the spacing and the strength of the clips shall prevent contact with neighbouring components, decreasing the abrasion risk or other damage during the life of the product.

3.1 Clip classification

The clips are classified in the following types, based on their main feature:

Omega clips: As the name suggests, they are clips with omega or double-omega shape, which can be used as distance clips or when there is not too much packaging space. They are the simplest ones.



Figure 4. Omega clip. Obtained from www.jetpress.com

M-clips: Their main feature are the devices (wings) with M shape that holds the pipe in position, allowing to easily reach assembly and pull off requirements.

Clips with locker: The locker can be implemented in a clip in situations of packaging lack, high pull-off force or highly vibrating and movements zones.

NVH clips: Some of the previous clip types could also be included as NVH clips when they are combined with rubber. The rubber addition shall reduce the vibration and noise in some critical parts of the vehicle.



Figure 5. NVH clip. Obtained from www.jetpress.com

3.2 Requirements

In structural terms, the clip must fulfil technical requirements. The research is focused in ergonomic and safety criteria. As mentioned in Chapter 1 Introduction, the assembly force of the line into the clip shall be low enough to avoid strain injuries, and the pull-off force for remove the line from the clip shall be high enough to stay in its original position avoiding risks of any kind.

Table 1. Clips requirements.

Requirements	Ergonomic criteria	Safety criteria
Assembly Force [N]	< 50	---
Pull-off Force [N]	---	>35

3.3 Models

The following clip models are chosen for the research object of the project, as the most relevant:

- Old design model: Clip 31405437
- New design model: Clip 33319355-01/12
- Redesign model: New clip re-designed based on the 33319355 clip model (see Chapter 4, Section 4.3.3 Redesign)

The responsibility of the studied clips belongs to the department 97424 of Fuel lines and EVAP of Volvo Car Corporation, that together with the supplier are responsible of its control, testing and redesigning in case that could be necessary.

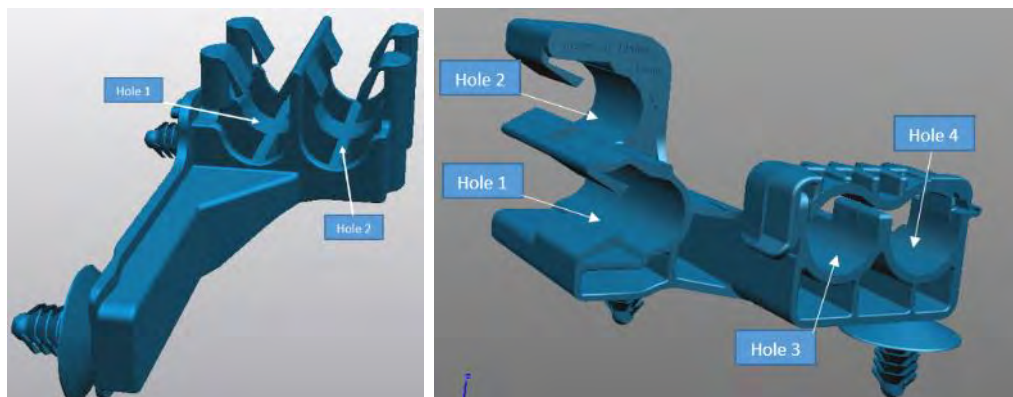


Figure 6. Left: Old design model, Right: New design model; Obtained from Teamcenter software

3.3.1 Specifications

In the above Figure 7, it is explained the notation that will be used in all the project. Holes 1 and 2 for the Old design model, and holes 1, 2, 3 and 4 for the New design model.

It is remarkable, as explained in Chapter 1, Section 1.3 Objective & scope, that the study is based on assembly and disassembly forces of the holes 1 and 2 of both models. Additionally assembly forces of holes 3 and 4 of the New design model are also studied.

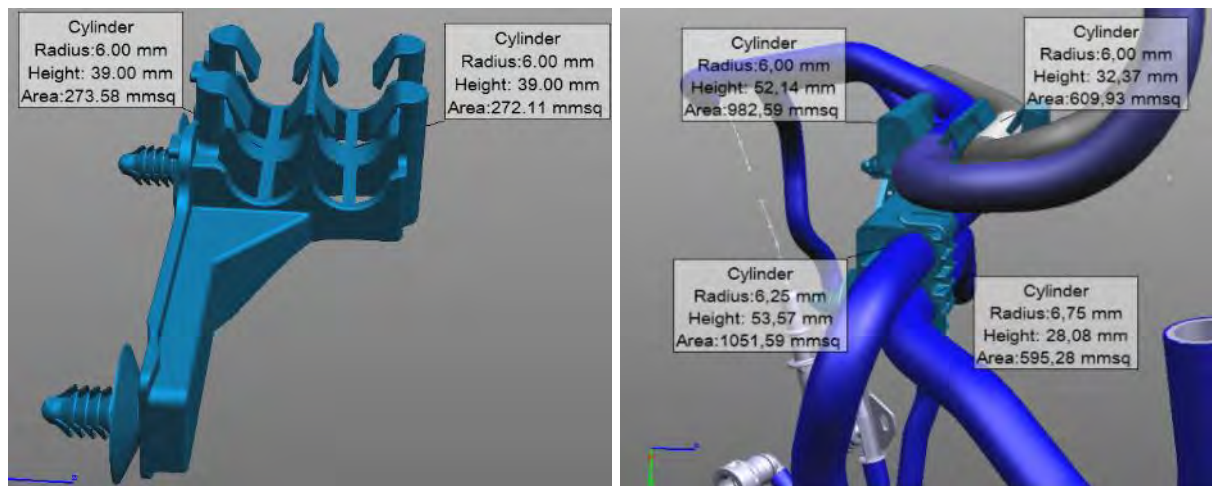


Figure 7. Clip measurements. Left: Old design model, Right: New design model; Obtained from Teamcenter software

The Table 2 summarizes the dimensions of the tubes retained in both clip models.

Table 2. Diameters of tubes hold in the clips

Hole	Tube diameter [mm]	
	Old design model	New design model
1	12	12
2	12	12
3	---	12.5
4	---	13.5

The following image shows the variant PV526746VSOPT19V (project 517B), where the four holes are in use.

Note that the hole 4 is holding a pump that does not belong to the fuel lines system, but to the vacuum system of brakes, of the Steering & Brake department.

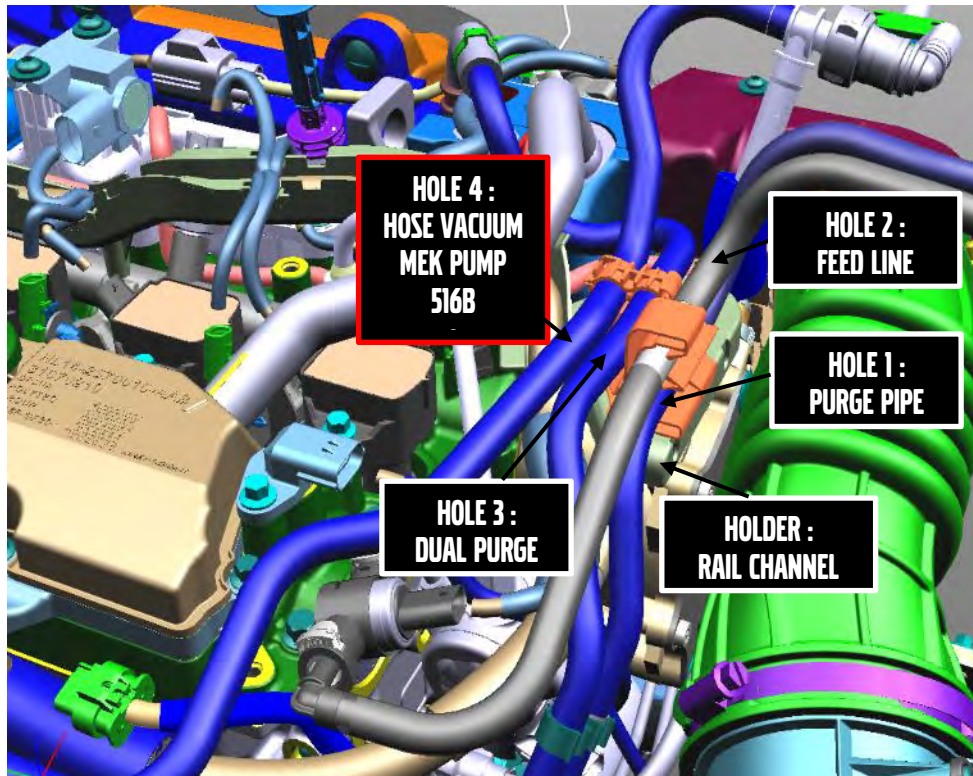


Figure 8. Overview of clip model in the car; PV526746VSOPT19V (project 517B)

3.3.2 Material

The material of both clips is Polyamide 66 (PA66), an important parameter that will significantly affect the performance of the real tests and simulation. The Polyamide 66 has different behaviour depending on the conditions in which the clip develops its life cycle.

It is a fact that a Volvo XC90 clip does not have the same behaviour in Kiruna, Sweden, than the same XC90 in Dubai, also taking in mind the different situations that may occur in the engine compartment. Therefore, the humidity and temperature conditions of the material will be critical parameters in the study.

It is important for the reader to take into account that the material used in the simulation process is an specific type of Polyamide 66, called *Scanamid 66_A331E* from the company Polykemi (Polykemi, 2017).

Table 3. *Scanamid 66_A331E* properties

Density [g/cm ³]	Yield strain [%]	Strain break [%]	Tensile Strength [MPa]
1.11	35	>50	<60

The material values specified in the Table 3 above, are referred to conditions at room temperature (RT) and 50% of relative humidity (RH) (Material data center, 2017).

4 Clip simulation

This chapter contains the development and explanation of the clip simulations carried out in the research project, so a simulation overview, the Abaqus structure and the simulation iterations are included.

4.1 Simulation overview

The reader will find a brief summary of the different functions that are performed to simulate. The simulation complexity depends on the type of analysis, the geometry, the different parts that make up the body, the mesh, the loads that are applied onto it and all the external conditions in which the simulation is carried out.

It is noticeable that the following three steps could be embedded in different simulation software as modules, or on the contrary could take part of the same simulation software:

- Pre-processing
- Computer Solving
- Post-processing

4.1.1 Pre-processing

The first step is to obtain the geometry file and there are two options to get it. Firstly by importing it from CAD design program, since the pre-processor software is already prepared to read the CAD format. The second option is to create the geometry in the same pre-processor software. This last option is interesting for the purpose of making design improvements and see how the results of the simulation changes in a single software.

When the CAD geometry is imported, it has to be checked and some errors could be detected, such as little variations of the original shape or small gaps, which means that the file has not been successfully read at all. Because of that, it is important to carry out the clean-up of the geometry. During the clean-up these problems are fixed (most of the times in an automatically way) so that the geometry is ready to be used.

At this point, it is important to make distinction between 2D and 3D model. Sometimes, when a 3D model has low thickness, for instance a 3D sheet, it can be simplified as a two-dimensional plate, since the complexity of the simulation decreases while the behaviour of the geometry gives similar results. For this study a three-dimensional model is used.

A shell mesh or surface mesh has to be created firstly. Triangle elements, quadrilateral elements, or a mix of both elements could form the shell mesh. Different parameters as maximum element length, minimum element length or angle have to be set in the software to create the requested mesh. Once created, it is necessary to check possible errors and improve the mesh in critical parts, such as holes, flanges or wings if necessary.

A high quality surface discretization means a greater chance of obtaining a good volume mesh, because the latter is created from the previous surface mesh. It is performed again a checking and improvement process but in this case for volume elements.

External conditions, loads, boundary conditions, etc. are applied in this step. One can define almost infinite parameters, such as type of analysis, speed, temperature, necessary outputs and more. Then, proceed to save the working file as *.ans*, and export the mentioned one with an available format for the solver (*.inp*).

4.1.2 Computer solving

One of the most important settings for the solver arrived to this point is the type of analysis. Abaqus has two different solvers, Standard (default) and Explicit (see Chapter 2, Section 2.2.2 Linearity, nonlinearity and analysis type). In this project nonlinear structures are studied, as it is detailed in the Chapter 2, Section FEM in Abaqus and Dynamic analysis is the chosen option that means Abaqus/standard is the used solver.

Abaqus/Standard receive the *.inp* file exported from the Pre-processor, as the input file containing all the necessary information for try to solve the problem. After a variable running time the user receives an info package with the following relevant files:

- File *.pre*: Details the pre-analysis that the solver performs before starting the first increment, checking possible issues in the mesh, elements, parameters or inconsistencies between them.
- File *.odb*: Binary output file, ready to be read in the Post-processing, that contains the analysis result.
- File *.dat*: Text output file that contains results.
- File *.msg*: Detailed progress of the analysis and its iteration process.
- File *.sta*: Statistics file that summarizes the analysis information in terms of step, increment, iteration and time. Summary of *.msg* file.

4.1.3 Post-processing

This is the last step, in which all the result are visualized in a way that they can be easily interpreted. The *.odb* obtained from Abaqus is imported in a software called Abaqus viewer, which provides several visualization, contour or limits tools to display the results from the analysis in graphical form (Getting started with Abaqus: Interactive Edition, 2012).

In Abaqus Viewer one can visualize all the outputs that the user has previously asked for in the output. For this study is essential to represent the reaction forces in Y axis, strains and the energy balance with the kinetic energy and external work in order to check accuracy of the quasi-static results (see Chapter 4, Section 4.3.2.14), and plastic strain contour plot of the model seeking check the behaviour of PA66 (Polyamide 66) material.

4.2 Abaqus structure

First, it is crucial for the reader to understand the language that Abaqus uses and which is the main structure of the software, since the Pre-processing will be done with a different software, ANSA. This software will translate the data to a “readable by Abaqus” file.

All the models are formed by a node cloud, and each node is identified by its coordinates and Id number. The different elements are associated with a specific number of nodes (depending on the element type) and the adjacent elements share some of their nodes.

All the forces, displacements, constraints, etc. are applied on the nodes, but the requested outputs can be from the nodes or from the elements, depending on the required results.

Before establish the loads or BC into the model, it is important to define the Sets. A Set is a group of nodes or elements which are identified and saved in the program, so one can use this group of nodes for any purpose without selecting manually again. It is good to give descriptive name to the sets, because the number of sets is usually big. Of course, it is possible to apply a load on one or more nodes without using a set, but it is highly recommendable to have previously defined the sets before making any modification on the model.

Once the sets are defined, it is time to apply the loads or the boundary conditions on them, and it is important to be aware that all the BC must belong to a certain Step.

The loads, concentrated or distributed, requires the direction and the magnitude (in this study the units has been previously defined, and the forces are measured in N).

Regarding the boundary conditions, it can be defined as displacements, velocities, accelerations or fixations. They are applied on those regions where exit known conditions. These BC requires the magnitude and the degrees of freedom (DOF), which are divided in translation (1, 2, 3) and rotation (4, 5, 6).

In order to define the type of analysis, there is need to create a Step. The Step helps to define a specific action during the simulations, since sometimes the model needs to represent different actions. For instance, in the clip simulation, each assembly or pull-off force will be linked to a certain step in specific model, so that the programme can carry out the actions during the right moment. It is important to notice that using more than one step in one simulation is possible when using static analysis, in this case will be used a Dynamic quasi-static analysis so, in this study each file contains one step, which represents one action. Parameters within the Step will be defined:

- Type of analysis.
- Time settings.
- Requested outputs for Post-processing analysis.

Summarising, the steps contain all the parameters related to the simulation. All the forces and BC have to be linked with one of the Steps, and each of the forces and BC should be linked with each Set. Of course, different Sets can be shared by different loads and BC, and also by different Steps.

Contact is defined as the interaction of one part with a different one or with itself. It is possible to implement different types of contact in Abaqus software using automatic or manual tools. A contact means also the need to define friction values and interaction between surfaces.

As an important aspect for the study, it is remarkable that Abaqus is not a built-in system of units, so before starting with the model there is a need to define and establish the system of consistent units that are going to be used. In this case, it is based on the International system of units, but with the length unit in millimetres (Getting started with Abaqus: Interactive Edition, 2012).

Table 4. Units of the project.

	Length	Force	Mass	Time	Stress	Density	Energy
Unit	mm	N	tonne	s	MPa (N/mm ²)	Tonne/mm ³	mJ (J 10 ⁻³)

4.3 Simulation iterations

In this section one can find a pre-study and the simulation development of the new and old design models.

4.3.1 Pre-study

It is important to emphasize some points before going into the explanation. The best way to understand the model and its behaviour is to take small steps and build the model slowly. The reader should understand the complexity of a model with contacts, friction and nonlinear geometries, where it is easy to obtain a non-converged solution. The idea of step by step mentioned before makes easy the localization of the errors in case some occur. Moreover, in order to obtain a good knowledge of the different software parameters, several comparisons are carried out. This chapter explains some of the most important differences and its effects on the results.

The simulation process begins with the realization of simplified models. To do this, it is necessary to select some clips with simpler geometries than the two mains, and reduce the complexity of the model by applying simple loads and simple boundary conditions. Although the results obtained from these simulations are not relevant for the study, it is important that the user get used to work with the software in these conditions and with this methodology.

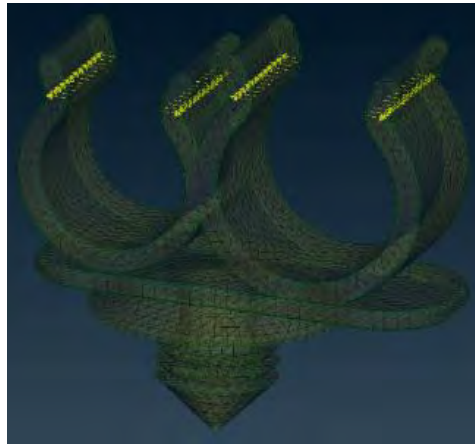


Figure 9. Basic model with loads; Obtained from ANSA software

In Figure 9, one can see the simple geometry of the clip, and the load applied. This load is the simpler option that could represent the behaviour of the insertion or removal of the fuel line, and it consists on applying a displacement on each part of the wing that corresponds to the radius of the fuel lines. It means that it has been applied the displacement needed so that the fuel line can get inside or out. There are four groups of loads, and each one represents the displacements applied on the correspondent nodes.

Some other simulations, similar to the previous one, has been carried out in order to establish the first parameters and get used to those type of simulations, however, the results are not significant, so this study does not go into detail with them.

4.3.2 Models development

From the point of view of the authors it is appropriate to perform some parameters-changing studies and check their differences in order to understand the behaviour of the software and establish the most suitable parameters for future simulations. The section contains the following modules representing the most important actions performed during the project, based on changes in the new and old designs. Explanations in this section are within the pre-processing context and are developed with ANSA software.

4.3.2.1 Topology and resolution

As the first step of a simulation, the topology and resolution settings are established, they must be consistent with the dimensions, the level of detail and the tolerances of the CAD file.

The topology involves the connectivity with neighbouring faces of the CAD file. Faces are connecting through their common boundaries and this process is controlled by Tolerance settings. There are four different modes: draft, middle, fine and extra fine. Each one specifies a value for the matching distances between the hot points (nodes on CONS representing their end points) and the curves on surface (CONS) . The fine tolerance mode is considerable accurate enough for this study.

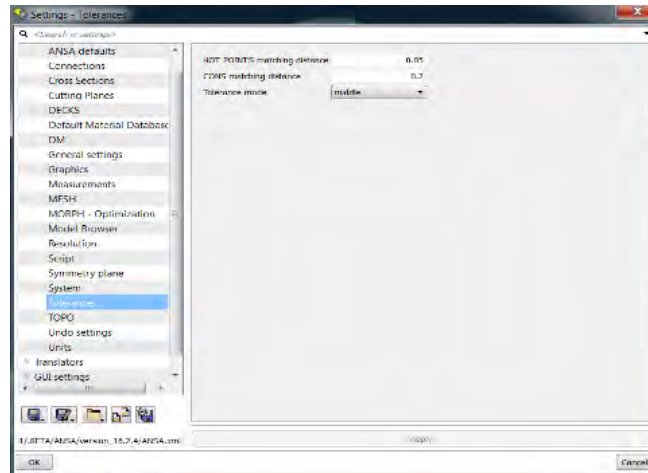


Figure 10. Settings window, Tolerances; Obtained from ANSA software.

The resolution controls the visualization of the 3D curves and CONS. A smaller CONS resolution (Perimeter length) allows a better perception and visualization of the curves in the model.

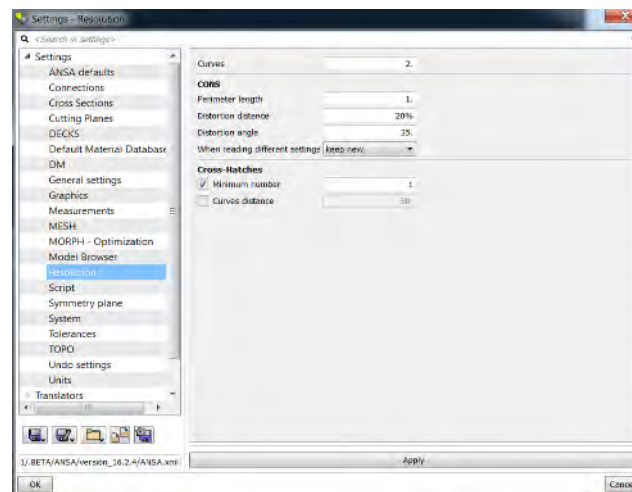


Figure 11. Settings window, Resolution; Obtained from ANSA software.

It is extremely important to always check the CONS perimeter length with the Target element length (meshing parameter). That last parameter is automatically taken from the resolution, therefore if the user changes this parameter has to be aware that the curves on surface perimeter length must be always smaller than the mesh target length and that will avoid problems in the meshing generation.

$$CONS \text{ resolution (Perimter Length)} < Target \text{ Element Length} \quad (4.1)$$

4.3.2.2 Importing

The CAD file is imported to ANSA software. A wide range of file formats are allowed. For this study .CATPart file from CATIA is the format used.

4.3.2.3 Check & clean up

Old design

After start the meshing procedure the model is checked and Cleaned-up. Some problematic areas may appear in the model, even more if the geometry is complex. ANSA software allows to check the geometry with an automatic check tool. This tool also allows to perform inspections of several parameters and features in the model, and will be used later on in this study.

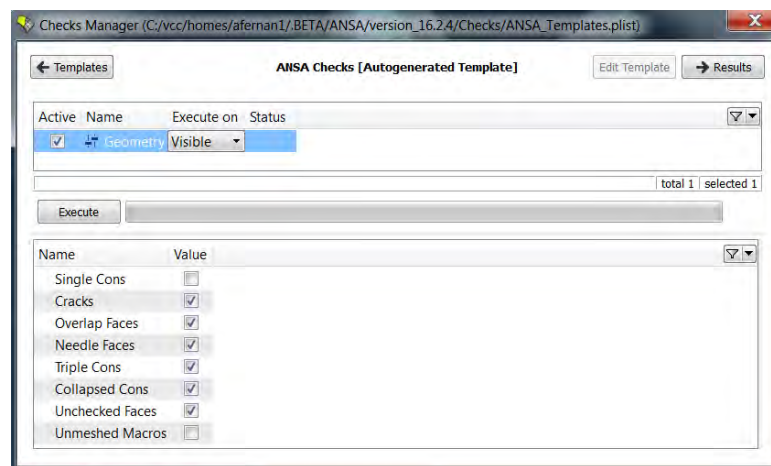


Figure 12. Check tool. Geometry; Obtained from ANSA software.

For the old design model no clean-up is necessary. Issues as Overlap faces, Needle faces, Collapsed Cons does not appear.

New design

The New design has some issues that cannot be fixed automatically, thus sometimes, fixing them in that way could cause a bigger inconvenient than one tries to avoid. Figure 13 shows the manually clean-up.

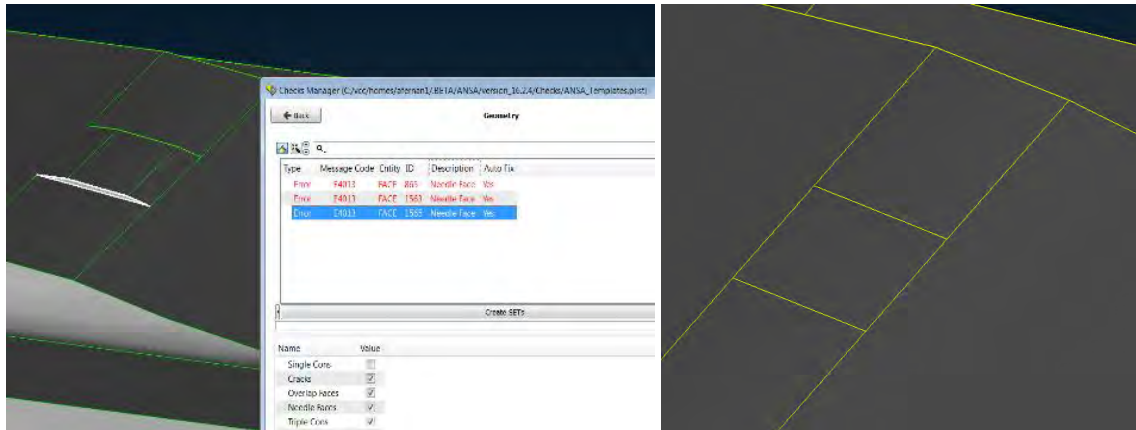


Figure 13. Manually clean-up of new design. Left: before clean-up, right: after clean-up; Obtained from ANSA software.

4.3.2.4 Defeaturing

One should consider check the model visually in order to improve features that are not detected in the check tool but could cause issues anyway. It is common to dispense with features that are not relevant in the model's behaviour. The logos are a clear example of non-necessary features for the study and which could cause issues during the pre-processing and in the simulation.

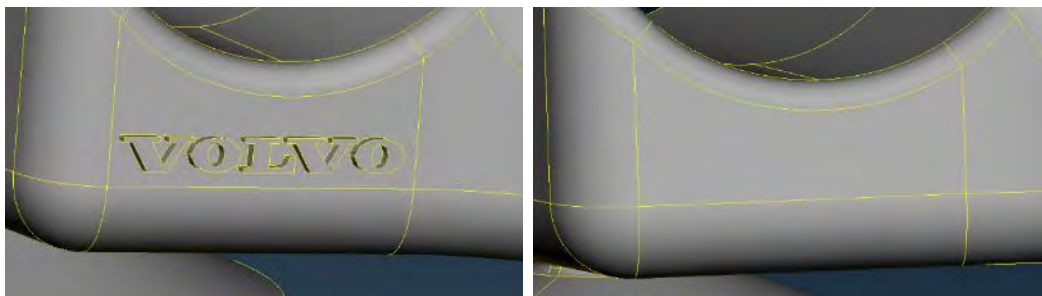


Figure 14. Logos defeaturing; Obtained from ANSA software.

The feet are the region that connects the clip to a certain part of the vehicle. The forces that appear in this region does not take part of this study, so the feet have a simple purpose of fixation. In several times, the feet have been meshed and the model presents convergence issues due to them and the solution could not be reached. It is normal to obtain those issues, since the foot is formed by excessively thin parts and really complex shapes.

It has been decided to avoid those issues by removing the feet, making an approximate and simplified model, which has a similar behaviour. It does not only avoid the problems, also decrease significantly the running time, since there is no need to have an extra fine mesh on them. This action, once it has been realized that it is a good alternative, is implemented in all the models studied in this project. On Figure 15 is shown how the simplification is done.

To carry out the defeaturing, specifically that one, or any other that request modifications on the CAD file, there are two options; make the modifications with ANSA or modify the file on CATIA and then import again the workpiece and start the pre-processing and mesh again, which could take more time. Due to it, anytime the piece do not requires really complex modifications, it is recommendable to use ANSA. In our case, there is only need to trim and cut the feet.

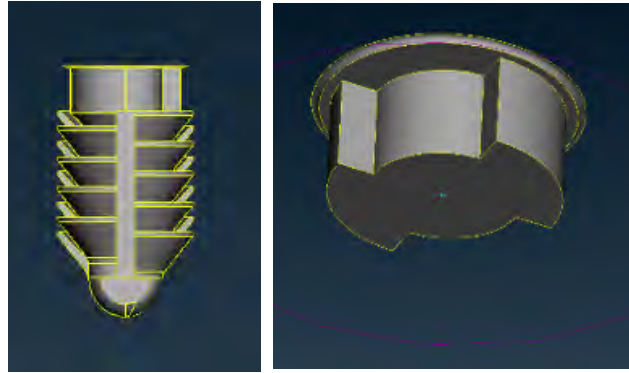


Figure 15. Feet defeaturing; Obtained from ANSA software.

4.3.2.5 Mesh

The mesh is the common feature of all the FE analysis, it is the way of divide a complex and impossible geometry to study into a known number of discretized elements easier to analyse. There are different meshing options within the ANSA software and two ways of meshing a model: Manually, or automatically with a batch mesh manager.

The batch mesh tool is a fully GUI -Graphical User Interface- driven tool which performs automatic mesh generation on the geometry trough customizable meshing sessions (ANSA version 15.2.x User's Guide, 2015, 969 pp.). This automatic option allows deciding between creating a new automatic mesh and loading a specific mesh scenario file previously created.

The other option is using the mesh toolbar and mesh the model part by part, which could take some more time, but also allows a high control and supervision of the mesh being made.

Due to the experience, it has been noticed that the best option is to make a general and not really finer mesh in an automatic way, and then re-mesh the critical parts of the model with finer mesh. Those critical parts refers to; points where there are strains concentrations, since those elements will have higher deformations, and if the elements are too big the solution could not converge; all the surface that will be selected as contacts, since the surface for the contacts needs small elements to avoid penetrations between them; and tricky and complex areas, since even if the strains are not really high, the elements will experiment high distortions.

In any case, to obtain a good mesh the user has to define the following aspects in which it is based on: The mesh parameters and the quality criteria. Those settings establish some standards that have to be fulfilled automatically, so they seek reduce to a minor number the manual improvements. The parameters has a default value, but the user can change them.

The 2nd order mesh is used for high displacements of the nodes and high distortion of the elements. In this study is necessary to use 2nd order mesh. To check it, the old design have been performed with both type of meshes. The models show different results and the elements of the 1st order mesh are highly distorted, so it is proved that this models requires 2nd order mesh.

There is need to take into account the convergence of the mesh. The convergence of the mesh (do not confuse with the convergence of the solution) refers to the biggest size of element allowed to ensure an adequate and accurate solution, since big elements could cause smaller reaction forces, von Mises stresses or strains than in the reality. To check that convergence, one should start with a big mesh size, note the results, and make some more simulations with finer meshes. When the results are the same for different mesh sizes means that the convergence has been reached. The first try is made with an element size of 1 mm. In some cases, and depending on the complexity of the model (contacts, BC and so on) the solution could not be reached because in some critical areas the mesh is not finer enough. The next simulation are made by re-meshing the critical parts, with element sizes of 0.7, 0.5 and 0.3. The results show the same values and same behaviour, so it can be assumed that the mesh is converged for those values.

Another important thing to take in mind is that there are two types of mesh, the surface mesh and the volume mesh. The surface mesh is the first that needs to be done, and on it are made all the modifications and re-meshing. The volume mesh is made from the surface mesh, and it is usually made just with the automatic tool.

Related with the solution file, the mesh could be a critical parameter. The greater the number of model elements, the larger the file size and computational time due to the amount of solved equations. A good engineer not only have to focus on getting proper and very refined results, but also on optimizing the available resources.

4.3.2.6 Properties and materials

The different parts of a model could have different properties and each one could define parameters as material, element type, colour and thickness of the specified part.

The material is the most relevant one, where the user can define all the mechanical properties of the part. Materials are linked to the Properties container trough a material ID (MID). The desired material could be loaded from a data base or its properties can be typed (or modified if necessary) within the Material container.

For this study a data base obtained from the CAE department is used, taking from it the necessary material Polyamide 66 (PA66). The Polyamide data (Density, Elasticity, and Plasticity) applies to a material conditioned at room temperature (RT) and 50% room relative humidity (RH).

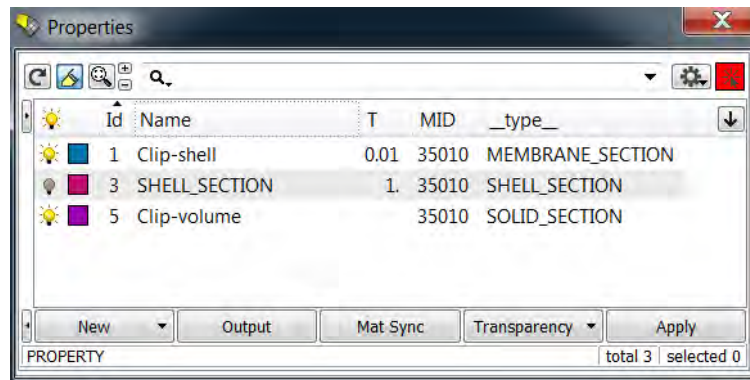


Figure 16. Properties container; Obtained from ANSA software.

Abaqus has a big library of elements, of which the most important are explained in the Chapter 2, Section 2.2 FEM in Abaqus. The surface mesh is defined as Membrane 3D element type (M3D) in order to allow visualize the results, as could be strains, in the solid mesh. The volume mesh of the clip needs to be defined as a solid type element, for that the user specifies C3D (Continuum 3D element type).

Thickness property is just relevant in element types as Membrane or Shell. In which a small value is used to affect as less as possible the Solid element results.

4.3.2.7 Boundary conditions

Boundary conditions are understood as all the parameters related with applying displacements, velocities, accelerations or fixations to the nodes of the model. Following is explained three parts of the model in which are applied different BC.

For an easier understanding of the following information, the Figure 17 represents the coordinate axis for Ansa and Abaqus software.

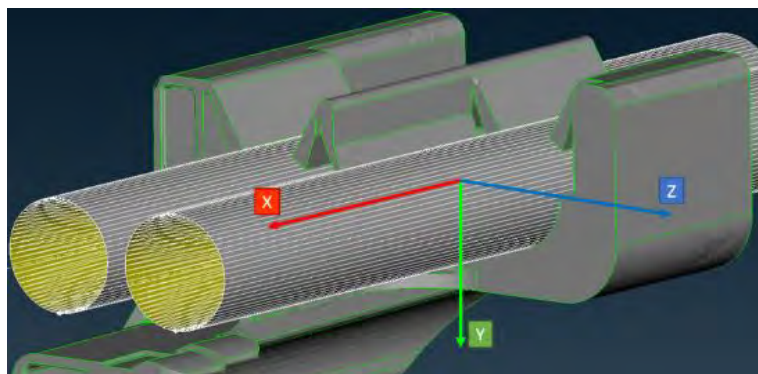


Figure 17. Abaqus coordinate axis; Obtained from ANSA software.

Tube – Old design

As mentioned in the following section, the whole tube, built as discrete rigid body or as analytical rigid body, is linked with a reference node. So each time the user wants to apply anything on the tube, there is just need to apply the BC to that reference node, so all the nodes will behave as that reference node.

In our model, it is assumed that the tubes only move vertically, up or down depending on the test, and the horizontal or transversal movement are not consider, since it is supposed that is the same on the real assembly line and also on the real test. Any of the rotations on this in the tube is neither consider.

As explained before, the contact between tube and clip is contact pair, as this has to be considered when applying the BC, since this contact only made that the tube does not penetrate into the clip. When applying those conditions on Abaqus, there is need to locate the initial position of the tube as closer as possible to the clip in order to save running time, so the total distance of the displacement is around 13 mm. After several simulations, it is noticed that the maximum reaction force is reached after 6 mm of displacement in assembly, and 7 mm in pull-off, so if the user wants to save more solving time and only focus on the behavior during the maximum force, the total displacement could be reduced. The simulations done in this study has different displacement depending if there was any issue during the simulation. Some of them has 13 mm, other 10 mm, or even 8 mm.

Tube – New design

During the performance of the real test it was realized that the lines rotates a bit during the insertion, so the new clip design model is improved by allowing the free rotation on X axis. That is, not adding any restriction on the rotation of that axis. Despite this, the reader should take into account that the rotation of that tube will increase considerably the running time, and in some cases will lead not achieve convergence in the solution due to complexity in the relation with the friction. It is necessary first to perform the simulation allowing rotation and check if it works.

Auxiliary tube – Old design

During the simulations on the old design, it is decided to use the same contact as is used for the tube, contact pair. With that contact, one could think that there is no need to apply BC, but to ensure the good behaviour of the auxiliary tube, the transversal axis, X, is fixed.

Auxiliary tube – New design

With more complexity in the model, it has been realized that the software did not respect the BC of fixing the tube in X, so when the force has too high, the auxiliary tube goes away from the clip, so the behaviour of the model was the same as if there is not any auxiliary tube.

To fix that, it is used the contact pair. In this case the tube is fixed to the clip, so the movement of the tube is governed by the movement of the clip. Because of that, with contact tied there is no need to apply any BC.

Feet

All the simulations carried out have the same boundary conditions for the feet, which are called “BC Standard” for a better understating of the following comparison. In this fixation, a high number of nodes are fixed in the X , Y and Z axes of translation, which means that the movement of the foot is highly restricted. Those nodes can be visualized on Figure 18, they cover part of the upper surface of the foot and also the external circle that is in contact with the metal sheet.

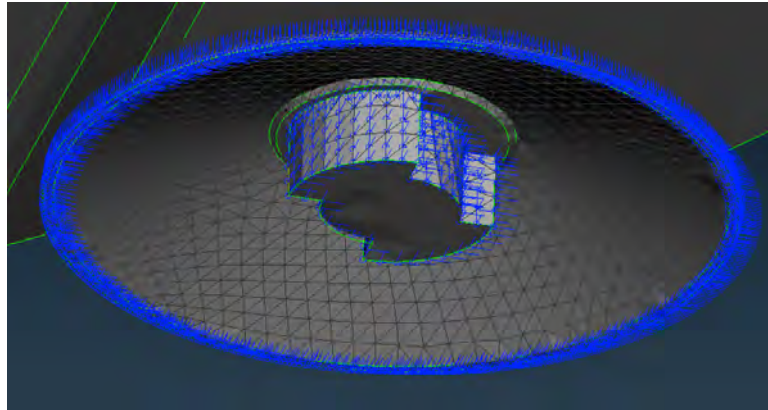


Figure 18. Standard BC; Obtained from ANSA software.

Three new ways of fixing the clips has been established in order to study which fixation could be the most realistic. All of them are less restrictive than the previous one in order to allow more movement in the clip and see the differences, and their representation is shown in Figure 19.

- **BC1:** Consist on fixing just a ring of nodes around the bottom part of the foot.
- **BC2:** It is the half of the first one. During the assembly only the lower nodes of that circle are fixed. On contrary, for the pull off force, the upper nodes are fixed. BC2 is less restrictive than the previous, since the upper elements on the foot are free to move.
- **BC3:** The last one is the less restrictive and only to nodes are fixed in each foot. As the previous, the nodes depends on which test is going to be simulated.

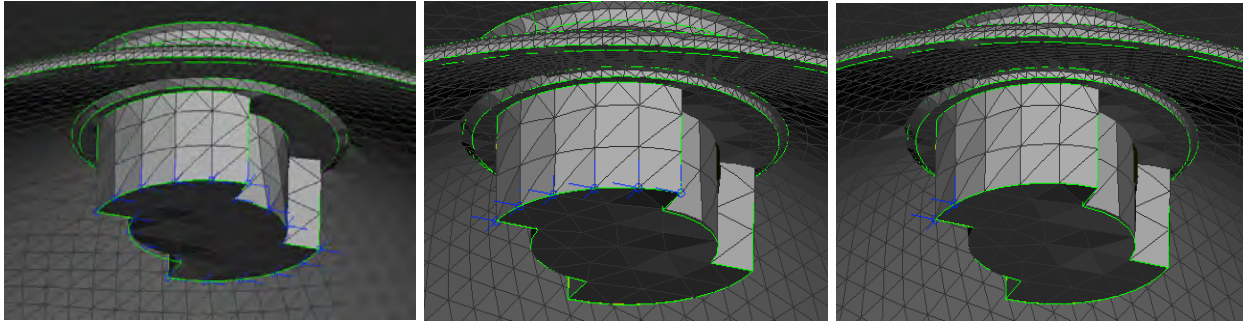


Figure 19. Left BC1, Middle BC2, Right BC3; Obtained from ANSA software.

To compare their behaviour, the assembly force of the hole 1 is simulated with those 4 different parameters, and the 4 graphs of the results of the reaction force are shown in Figure 20.

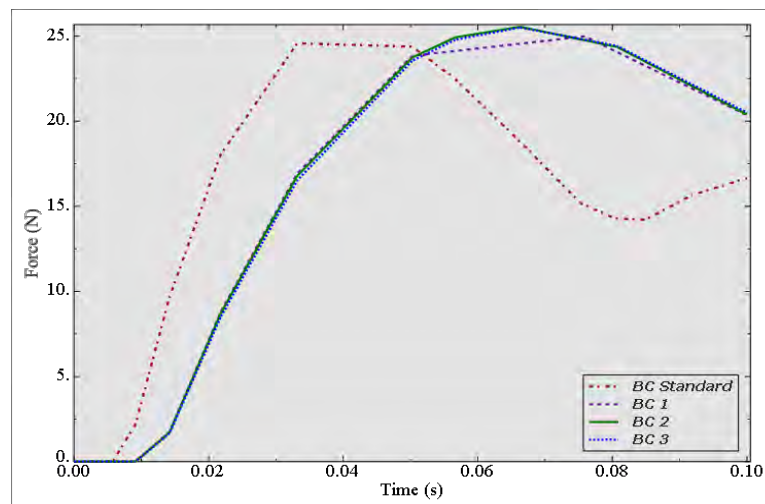


Figure 20. Boundary Conditions Comparison; Obtained from Abaqus viewer software.

To be able to understand the behaviour, there is need to highlight that once the entire real test have been performed, it can be appreciated that the clips bends down during the insertion of the tube and they bends up during the pull off force test. This slight flexion occurs because of the inaccurate fixation of the feet into the metal sheet, and this also happen in the real positioning inside the vehicle. That the clip bends during the tests means that the insertion or removal occurs a bit late than expected, and that it what can be appreciated on the graphs. The differences are that BC Standard model achieve the maximum force earlier, due to the higher fixation of the clip and the small change to bend. Also the way of fixing the feet is quite strong, and could not exactly represent the real behaviour of the clip during the test, but it can be accepted as valid since the results do not change.

The maximum assembly force achieved for all boundary condition models is almost the same, but there is need to take in mind that some differences can appear, which are not real, due to bad discretization could occur. It would be discuss subsequently. (see Chapter 4, Section 4.3.2.13 Output).

4.3.2.8 Tube

The whole model of the tube is developed in ANSA and not in CATIA due to the simplicity of the process and the tools available in ANSA.

During all the study, it is important to make assumptions related to the correlation of the real test and simulation results, trying to make the simulation as closer to the test as possible. Because of that, some conditions of the testing have to be considered, such as the material of the pipe that is used. In this case is used a steel pipe recovered with PA66. The elastic modulus (or Young modulus) of the steel is really high, around 200 GPa, and taking into account the low forces that are used, it can be assumed that the deformations on the steel are going to be negligible, so the young modulus can be assumed as infinite. In that case, the tube can be modelled as a rigid body.

A rigid body is a collection of nodes and elements whose motion is governed by the motion of a simple node (Getting started with Abaqus: Interactive Edition, 2012, pp 3-14), that simple node is known as reference node. In addition, it is advantageous for the solver, since it is easier and faster to work with rigid parts, instead of working with deformable parts.

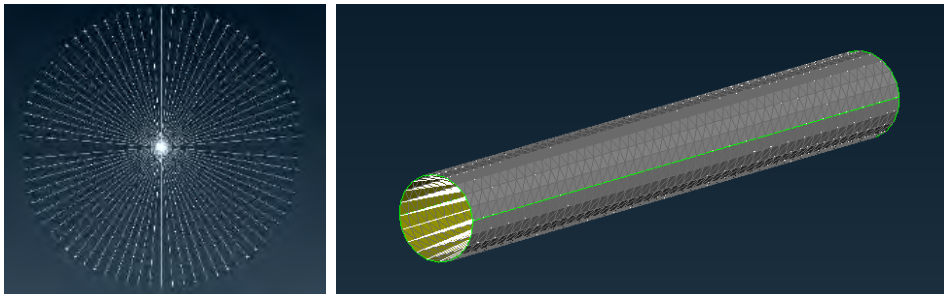


Figure 21. Discrete rigid body. Left: X-axis view. Right: Perspective view, Obtained from ANSA software.

The rigid body can be defined in the software as either a discrete rigid body or as analytical surface. The first one is obtained by meshing a body with elements and the second one revolving a two-dimensional geometric profile (Getting started with Abaqus: Interactive Edition, 2012, pp 3-14).

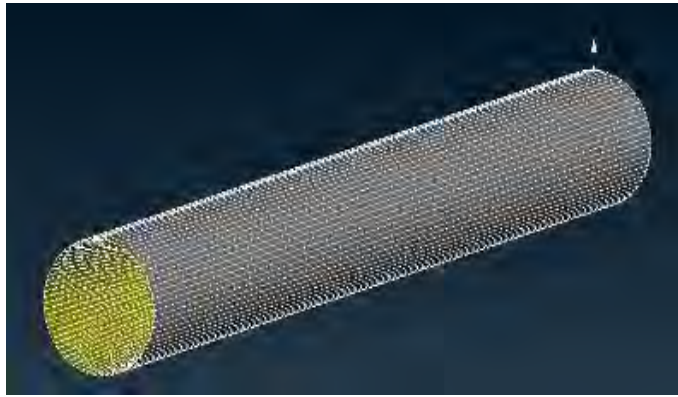


Figure 22. Analytical Rigid Body. Obtained from ANSA software.

Old design

Those two different ways of modelling are implemented in order to see the result's variation and the possible advantages and disadvantages:

- Discrete rigid tube
- Analytical rigid tube

Table 5 details and summarizes the data obtained in the assembly simulation for the Hole 1, it is important to emphasize that the clip model and the simulation's parameters are the same for both cases in order to obtain proper and accurate results.

Table 5. Comparison of discrete and analytical rigid tube. Assembly hole1 old design.

	Mesh	RF [N]	Max.Strain [%]	Energy Criteria	Running Time [s]	File Size [Gb]
Discrete rigid tube	Necessary	27,7	40-50	KE<WK	22145	10
Analytical rigid tube	Not necessary	27,7	40-50	KE<WK	6368	5

There is one criteria, related with the kinetic energy that must be fulfilled if the user wants a good behaviour on the simulation, and if it is not fulfilled the simulation can show strange and non realistic acts. The criteria refers to obtaining much more small Kinetic energy than External work (see Chapter 4, Section 4.3.2.14). The energy criteria is fulfilled in both cases, and the analytical rigid part is similar to a discrete rigid part in almost all of the aspects, having the same reaction forces (RF) and the same maximum strain (%), but with less computation effort, spending 3,5 times less computational time and the size file is one half of the discrete rigid tube file, which is really desirable. The reason why it has less computation time and a smaller *.odb* file is that the analytical rigid part is not necessary to be meshed.

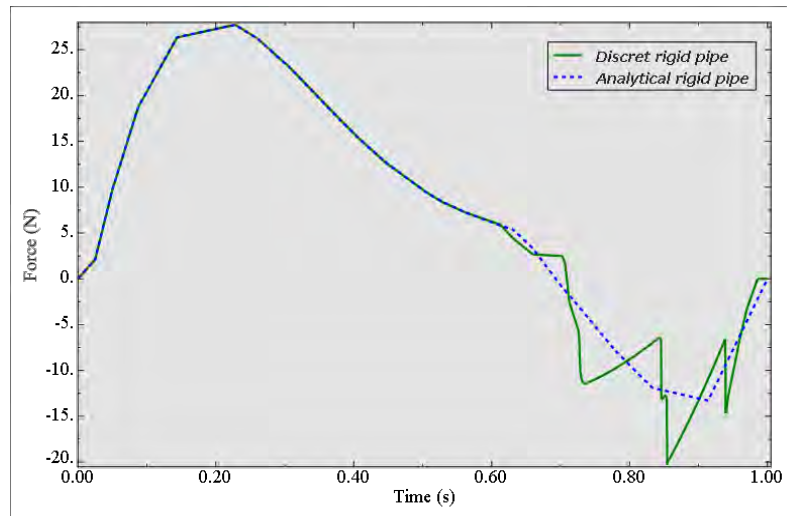


Figure 23. Comparison discrete and analytical rigid tube;
Obtained from Abaqus viewer software.

As shown in Figure 23, the analytical rigid pipe does not present noise, which is also undesirable, because it can cause higher or lower final values and could cause problems during the solution of the equations.

With the obtained results one can notice that the analytical rigid tube simulation is totally recommended, and there is no reason why using discrete rigid pipe.

New design

In the new design, all the tubes are modelled with analytical surface, and the discrete rigid body is not even used.

4.3.2.9 Contacts

The study of the forces needed to insert and remove a tube in the clip may not be reachable, or at least close enough to reality, without the contact module and the proper interaction between their surfaces. When two, or more, surfaces are in contact, a normal force appear on each one, and in case the contact has a friction coefficient assigned, also shear forces appear on them. Defining a contact allows to calculate those force to make a complete study.

Abaqus works as follows with contacts. The user will define both surfaces that are in possible contact, after that surface interaction models are created, where is necessary to establish some parameters as could be the *Scale factor*. Friction model will be also referenced in this surface interaction model.

There are different types of contacts. In this study, Contact Pair is implemented for the Old design clip, and after some undesired results, another type of contact called Contact Tie is added to the New design clip.

Contact Pair is the standard way in Abaqus to define that two surface are touching each other. As mentioned before the contact model is based on the interaction between the two surfaces. The first step is to define the potential faces coming into contact. For that, the solid facet of the desired parts are taken. Abaqus uses a pure master-slave contact algorithm. Master surface is defined as the surface that can penetrate into the slave surface, but not vice versa. The master is always the hardest part, in the context of this study it will be the tube, due to its rigidity as explained in the Chapter 4, Section 4.3.2.8 Tube. In this case, the rigid surface should be large enough to ensure that the slave nodes do not slide off and “fall behind the surface” (Getting started with Abaqus: Interactive Edition, 2012, pp 12-10). As the master is a cylindrical object, it is better for the behaviour of the contact calculation. Slave surface mesh size is refined enough to interact with any feature on the rigid-master surface and for not causing convergence issues due to large penetration of the master surface.

Contact Tie is a constraint used to fix together two surfaces for the duration of the simulation, for the analysed model it means that translational and rotational degrees of freedom are constrained. In the contact tie, the user defines the distance in which the slave nodes are tied with the master nodes (Getting started with Abaqus: Interactive Edition, 2012, pp 12-7).

Interaction between the surfaces in contact is a module defined within the contact tool. There are available different constrain enforcement methods within the interaction in order to help the software to solver the contact problem. This summarized in the following:

- Direct method: The solver tries to solve the problem without help from the interaction module, often achieving a non-convergence solution.
- Penalty method: Provides efficient solution, helping the software to avoid the penetration with a small sacrifice of accuracy.
- Augmented Lagrange method: Same as penalty method with a higher accuracy, due to more computational and time effort.

For the clip study a Penalty linear method is used, in order to obtain an accurate solution, but without spending long periods of solution process as could be with the Augmented method.

Old design

Specifically in the clip model, there is need to define contact between the tube and the clip, so that the tube can push the wings to get inside of the hole, and not cross the wings. A contact is also defined in the inner part of the wings for not allow them to penetrate in the clip.

For the first studied model, the Old design clip, a contact pair is used in wings and tube. Shown in Figure 24.

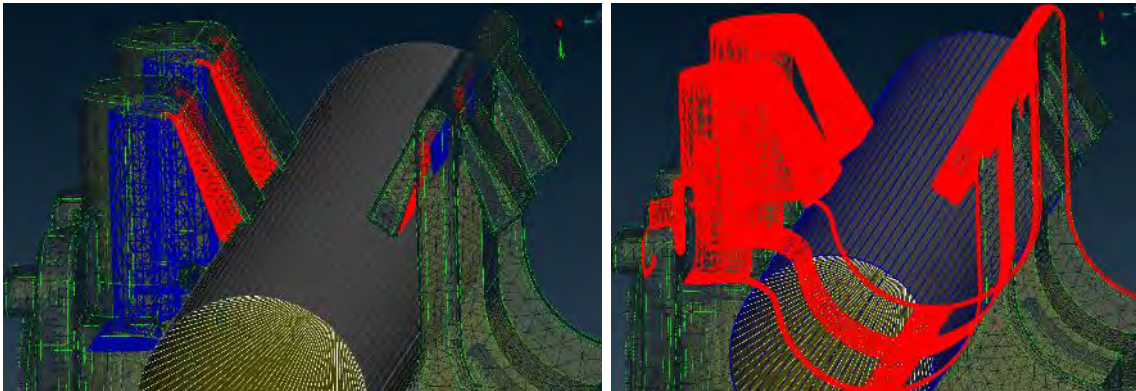


Figure 24. Contact pair studied tube. Left: wings contact, Right: tube contact. (Blue:master surface , Red:slave surface); Obtained from ANSA software.

New design

Some issues appear regarding the convergence and behaviour of the model with the contact pair for the auxiliary tube, as explained in Chapter 4, Section 4.3.2.7 Boundary conditions. Then, for the New design clip is introduced a new Contact Tie in combination with the original Contact pair for the auxiliary tube, as shown in Figure 25. The Contact pair remains in the studied tube, but in the specific case of Pull-off analysis the slave surface is reduced in the bottom part. This area is not being used in the pull-off, so it is deleted in order to save computational time, shown in Figure 26.

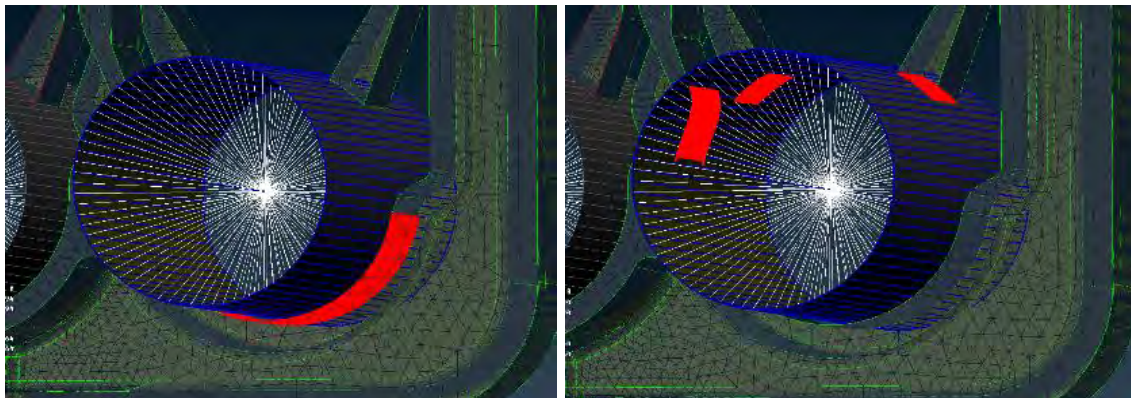


Figure 25. Contacts auxiliary tube. Left: contact tied., Right: contact pair (Blue:master surface , red:slave surface); Obtained from ANSA software.

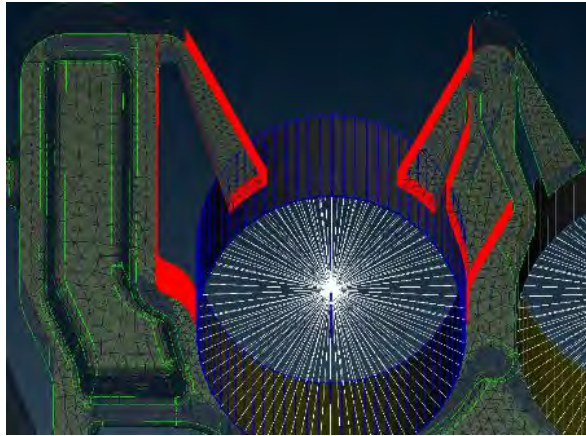


Figure 26. Contact pair studied tube (Blue: master surface, red: slave surface); Obtained from ANSA software.

During the definition of all types of contact, a surface interaction appears. The surface interaction module contains a critical parameter for the model, as it is the scale factor of the penalty linear method.

The scale factor can be also be known as *stiffness* scale factor, the user can change the default value of 0.1 in order to obtain a higher stiffness of the contact. Low contact stiffness may mean bad accurate results or even do not achieve convergence. As it is reduced the value of the scale factor the contact stiffness increase, the accuracy of the results could increase and the behaviour of the solver could be better (Imechanica.org, 2013).

For this study a scale factor contact comparison is carried out. The same model is studied with two different values, one is the default one (0,1), and the 2nd one is half of the default value (0.05).

After running the simulations, the obtained reaction force is the same in both cases. That means the scale factor of 0.1 is low enough (contact stiff enough) to obtain accurate results, but there is a difference of computational effort, shown in Figure 27.

ANALYSIS SUMMARY:		
TOTAL OF	409	INCREMENTS
	96	CUTBACKS IN AUTOMATIC INCREMENTATION
	2324	ITERATIONS INCLUDING CONTACT ITERATIONS IF PRESENT
	2324	PASSES THROUGH THE EQUATION SOLVER OF WHICH
	2324	INVOLVE MATRIX DECOMPOSITION, INCLUDING
	0	DECOMPOSITION(S) OF THE MASS MATRIX
	1	REORDERING OF EQUATIONS TO MINIMIZE WAVEFRONT
	0	ADDITIONAL RESIDUAL EVALUATIONS FOR LINE SEARCHES
	0	ADDITIONAL OPERATOR EVALUATIONS FOR LINE SEARCHES
	6	WARNING MESSAGES DURING USER INPUT PROCESSING
	215	WARNING MESSAGES DURING ANALYSIS
	0	ANALYSIS WARNINGS ARE NUMERICAL PROBLEM MESSAGES
	47	ANALYSIS WARNINGS ARE NEGATIVE EIGENVALUE MESSAGES
	0	ERROR MESSAGES
JOB TIME SUMMARY		
USER TIME (SEC)	=	1.16243E+06
SYSTEM TIME (SEC)	=	95603.
TOTAL CPU TIME (SEC)	=	1.25811E+06
WALLCLOCK TIME (SEC)	=	121890

ANALYSIS SUMMARY:		
TOTAL OF	235	INCREMENTS
	64	CUTBACKS IN AUTOMATIC INCREMENTATION
	1358	ITERATIONS INCLUDING CONTACT ITERATIONS IF PRESENT
	1358	PASSES THROUGH THE EQUATION SOLVER OF WHICH
	1358	INVOLVE MATRIX DECOMPOSITION, INCLUDING
	0	DECOMPOSITION(S) OF THE MASS MATRIX
	1	REORDERING OF EQUATIONS TO MINIMIZE WAVEFRONT
	0	ADDITIONAL RESIDUAL EVALUATIONS FOR LINE SEARCHES
	0	ADDITIONAL OPERATOR EVALUATIONS FOR LINE SEARCHES
	12	WARNING MESSAGES DURING USER INPUT PROCESSING
	158	WARNING MESSAGES DURING ANALYSIS
	0	ANALYSIS WARNINGS ARE NUMERICAL PROBLEM MESSAGES
	38	ANALYSIS WARNINGS ARE NEGATIVE EIGENVALUE MESSAGES
	0	ERROR MESSAGES
JOB TIME SUMMARY		
USER TIME (SEC)	=	6.73098E+05
SYSTEM TIME (SEC)	=	55900.
TOTAL CPU TIME (SEC)	=	7.28998E+05
WALLCLOCK TIME (SEC)	=	72143

Figure 27. Analysis summary. Left: scale factor 0.1, Right: scale factor 0.05.

The model with 0.05 (higher stiffness) needs half of the time, and half of the increments to obtain a convergence solution. So a stiffness scale factor of 0.05 will be used in subsequent models.

4.3.2.10 Auxiliary tube

Old design

After making some simulations of the assembly and pull-off forces of the tube with the contacts, the next step is to add an auxiliary tube in the hole that is not tested. This feature improves the model, making it closer to reality, and thus, gives more complexity and adds more equations to the solver, which could cause solution convergence problems and a larger solving time.

In terms of reaction forces and strain concentrations, having an auxiliary tube in the non-tested hole cause higher forces in the tested hole, since the movement of the clip is more limited and restricted. The strains also increase for the same reason.

In order to compare both models, it is important to quantify the increment of the force in a general way. Sometimes adding and auxiliary tube adds so many issues, therefore the solution can not be reached. In this case, an alternative option could be to approximate the model with the auxiliary tube to a model without it, but taking into account how much the results change.

In Figure 28, one can see the difference in the reaction force by adding or not the auxiliary tube, and in Figure 29 are represented the maximum strains of the models.

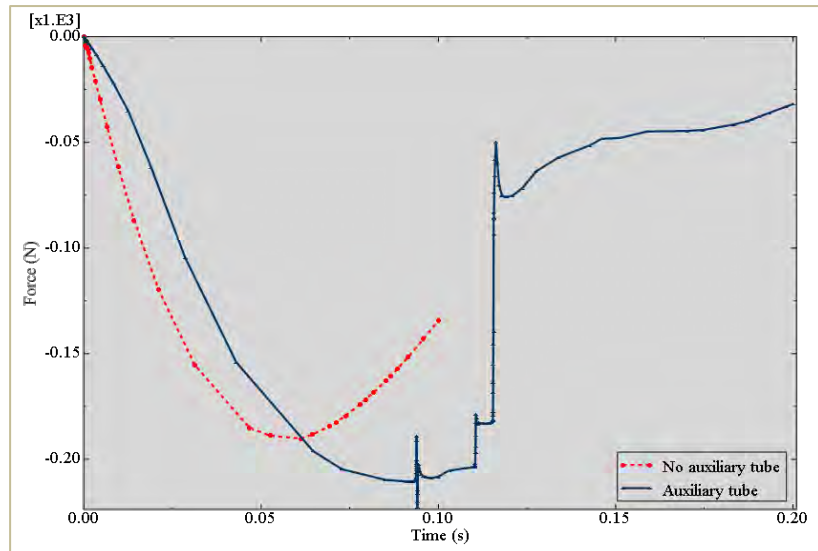


Figure 28. Comparison auxiliary tube. Obtained from Abaqus viewer software.

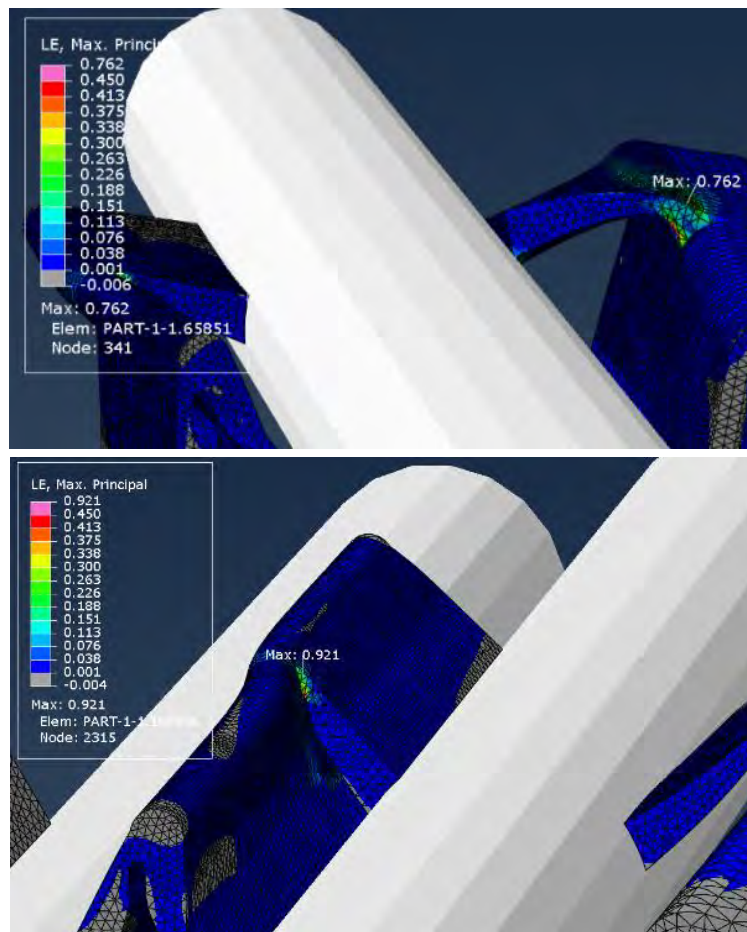


Figure 29. Strains comparison; up: without aux.tube, down:with aux. tube; Obtained from ANSA software.

4.3.2.11 Friction

In the situation that two surfaces are in contact, it is necessary to apply a friction interaction, in order to achieve a behaviour of the model as closer as reality. One should notice that friction value is established within the friction interaction.

The Coulomb friction model is the common used for contacting surfaces, and it obtains the shear stress for the surfaces in contact based on the following equation:

$$\tau_{crit} = \mu p \quad (4.2)$$

Where μ represents the coefficient of friction (between 0 and 1) and p the contact pressure between surfaces. (Getting started with Abaqus: Interactive Edition, 2012, pp 12-5).

The friction is one of the critical parameters which greatly affects the convergence in the solution. In order to study the possible differences achieved with different values both models Old and New design are implemented with different frictional values.

Old design

It is expected that the reaction force needed to assembly the tube increases with the friction coefficient between material and clip. To verify that and estimate the value of the increment, two models with the same parameters are simulated, one with coefficient value of 0.1 and the other with 0.32.

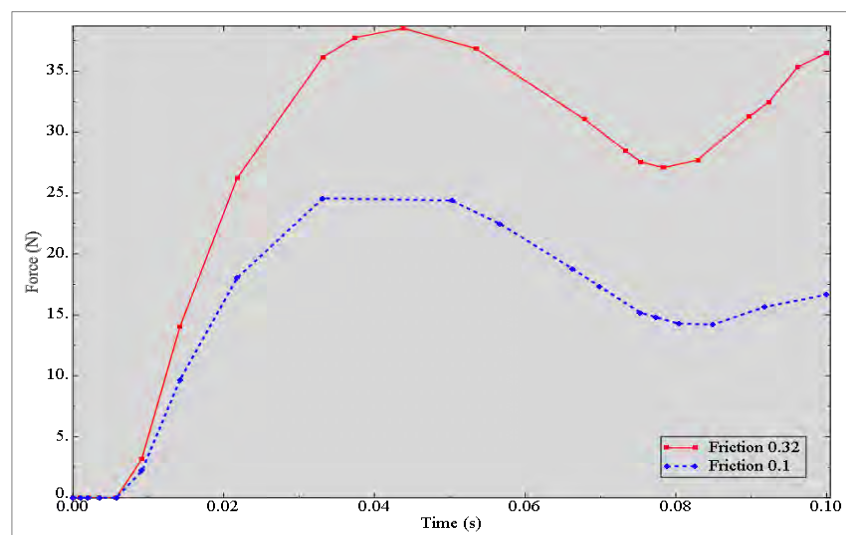


Figure 30. Comparison with different friction values. Obtained from Abaqus viewer software.

The results shows high differences on the reaction forces, but similar behaviour. In addition, one can check at the .sta file that higher friction coefficients values need more increments, and thus, more computational time.

New design

The study in the Old design shows a clear friction-dependency of the results, so the new model seeks to obtain its final results with the real friction values, 0.32 at hole 1 and 0.4 at hole 2. Anyway, reached this point, and due to the high complexity of the models, it is been decided to carry out some initial simulations with low frictions coefficients, in order to be able to localize the possible errors, as it was explained previously.

4.3.2.12 Velocity

The velocity during the test is establish in 100 mm/min, so the simulation should be performed in the same way. Taking into account the equation (4.3), the velocity can be set in two different ways.

$$v = \frac{d}{t} \quad (4.3)$$

The first one is applying the velocity as boundary condition, and then applying the step time, so that velocity and the time are establish, and the displacement is set with the value obtained by solving the equation (4.3). The other one is applying a displacement as boundary condition and the step time, taking into account that this relation gives the velocity. For this analysis is used the second alternative since it is needed to be extremely accurate with the displacements inserting or removing the tube.

Old design

In this case, two pairs of models are studied in order to compare the results of the maximum reaction force. In both occasions, two different velocities are applied to the same model, the requested one in the technical requirements and a higher velocity. One pair of models has a friction coefficient value of 0.1 and the other pair do not have friction.

Table 6. Comparison with different velocities.

Friction coefficient	Displacement [mm]	Time [s]	Velocity [mm/s]	RF [N]
0	13	1	780	21.06
	13	7.5	100	22.02
0.1	13	1	780	27.7
	13	7.5	100	28.1

The reaction forces with different velocities differ little. On the models with no friction, the difference is of 0.3 N and for the models with friction it is 0.4 N. As in the previous section, these small differences could be a result of the discretization or sampling, not a result of the differences in the velocities. That means that the sample representing the maximum value can be obtained in different time, and this could be the reason why the values differs.

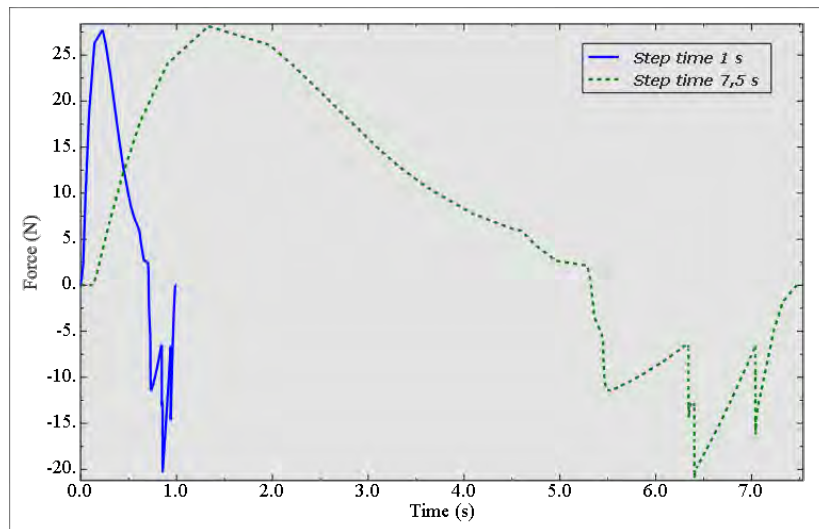


Figure 31. Comparison with different velocities ; Obtained from Abaqus viewer software.

In reaching to this conclusion, the two graphics shown in Figure 31, with different velocities has been compared. Even with the different time axis, it can be noticed that both graphics follow the same path and the variations are because of taking the samples in different times.

In this case, the higher value is the most accurate, and the maximum real force could be even a little bit higher. However, the conclusion about the different velocities is that the velocity does not affects the results, so any step time that works good can be chosen.

New design

With the new clip design, the complexity of the model has significantly increased, since some new parameters are added, trying to get it closer to reality. Because of that, the running time of the solver has also increased, reaching times around 30 hours. After several attempts trying to reduce it, it was noticed that decrease the step time is a good solution. The results show same reaction forces, same strains and same behaviour of the clip. So, all the simulations have been done with a step time of 0.1 seconds.

4.3.2.13 Output

Abaqus can generate a big amount of outputs. The user must specify the required outputs for interpret the results. Output parameters are defined within the Abaqus Step.

There are two different types of outputs available in Abaqus. Field Output is generated from data spatially distributed in the model, than can be visualized using the deformed shape or contour plots. History output is generated from data of specific points (Abaqus/CAE User's Manual, 2012, Section 14.4.2), as could be the reaction force of the node that governs the rigid body and give the assembly and pull-off forces.

When setting the requested outputs on Abaqus, there is need to establish an output frequency. It determinates how many increments are used to obtain one output sample or frame. As default, the value of the frequency is 10, a quite high value, so it is recommendable to decrease it due to the following reason.

For instance, working with a 150 increments file and demanding a history output frequency of 10 means that the results obtained show 15 samples, and in each one is represented the requested outputs. If working with the same 150 increments file and demanding a frequency of 1, the results show 150 different samples, each one with all the requested outputs. An output frequency of 1 can be translated as a higher results accuracy and a more representative graphic when plotting, but also as a higher output file size.

This parameters is really important and is needed to be careful with higher frequency values, since it can show unreal values, such us lower reaction forces as can be shown on Figure 32.

Below are shown the results of three different simulations, each one with the same parameters but with output history frequency values of 1, 5 and 10 (default setting) respectively. To carry out the comparison, the output variable studied is the reaction force in Y axis applied on the tube since that force is related with a single node and it is easy to visually interpret on x-y plots.

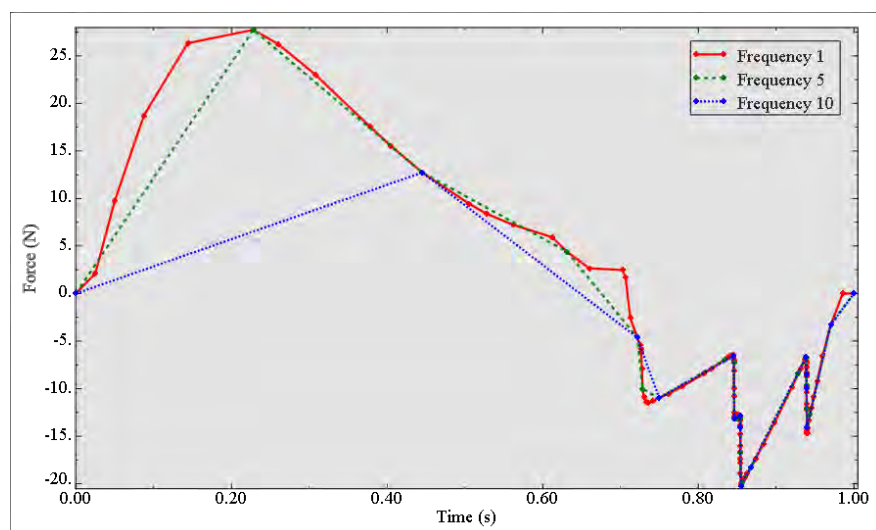


Figure 32. Comparison with different output frequencies; Obtained from Abaqus viewer software.

The results show different maximum reaction forces, while all of them should be identical. One can think that the simulations are improperly carried out, but the reality is that all the analysis are correctly done and the three has the same values. The problem occurs during the visualization and it is associated with a poor discretization of the values. Actually, when looking at the *.sta* information file, one can see that the simulation of the three analysis it is exactly the same. If the software takes high frequencies and therefore few samples of the reaction force, its representation can show

an unrealistic solution. Focusing on the maximum reaction force (most important value in this analysis), if the peak of highest values is in between of two consecutive samples, those high values are not collected on the results, as happen in the 10-frequency graphic. On the contrary, low frequencies, like 1, means more samples, which mean more realistic results and more representative maximum values. In between of those values there is plotted the 5-frequency graphic and its maximum value is the same as the 1-frequency. However, as shown on the graphics, this a random coincidence and the most probable situation would be that in other model this would not happen.

Another issue to highlight is that the increments become smaller as the simulation progress, so the differences between the results of different frequencies becomes closer. However, in this study, the force of interest (maximum force) is reached and the beginning of the simulation.

Summarizing, the output frequency influences the visualization and representation in the Post-processing of the results. The different calculations made by the solver are the same. It is strongly recommended to change the history output frequency to 1 for having an accurate final representation of all the calculations carried out by the solver.

4.3.2.14 Energy criteria as verification method for a quasi-static analysis

As explained before in this project, the contacts and friction of the model cause that a static analysis may not be enough to achieve convergence in the solution.

The Chapter 2, Section 2.2.2 Linearity, nonlinearity and analysis type explains that the dynamic analysis uses an implicit time integration to calculate the quasi-static response of the system. The advantage of this type of analysis is that it can be used to solve an unstable static analysis, whose solution does not convergence.

Otherwise, the inconvenience of this type of analysis is the results' reliability, since there is not Abaqus specific tools to check it. The results of the analysis will be accurate enough as long as the simulation is giving an adequate quasi-static response. Recent studies (Hammar (2012) and Abaqus, Quasi-Static Analyses (2005)) consider that an examination of the energy balance could provide a way of verify if the result reflects a quasi-static solution.

At this regard, the Kinetic energy should be small enough in comparison with the External work, in order to achieve the desired stability.

The following Figure 33 represents the resulting reaction force for one of the simulated models. The Kinetic energy is low enough, except in a peak at time 0.08 that is slightly higher. The maximum reaction force is achieved at time 0.06, so the possible inconvenient that this cause is not take into consideration. The possible instabilities arrives after obtain the desired result.

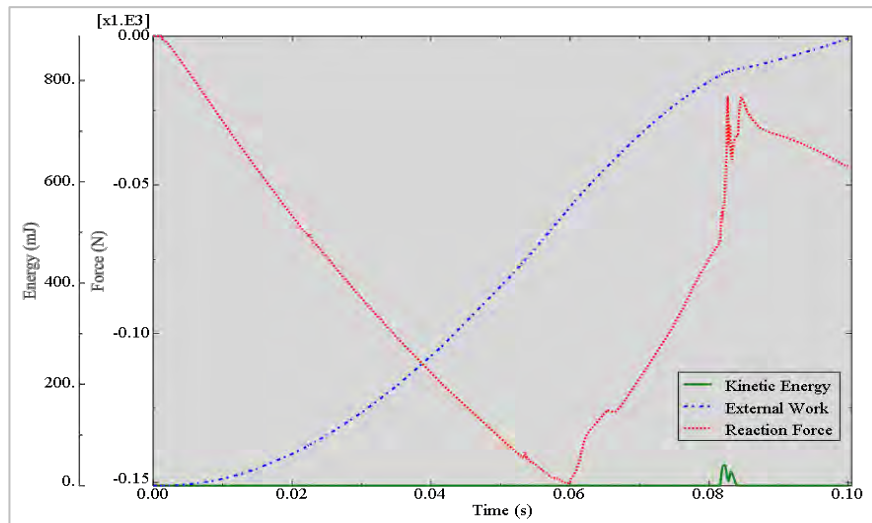


Figure 33. Energy criteria example.; Obtained from Abaqus viewer software.

One should also take care of, as an important parameter, the velocity of the simulation. As the speed of the model increases, the response of the part could become less static and more dynamic. Hence, the material velocities increase in relevance. In that situation one could check a significantly increment in the Kinetic energy, moving away from a valid energy balance and the quasi-static response.

4.3.2.15 Assembly 3 and 4

The New design clip contains a locker fixing the holes three and four. The holes with locker are characterized by achieving extremely high pull-off forces but low assembly forces. The locker allows the designer to avoid the wings that in most cases work as mechanical stop.

The pull-off force of the locker is not part of the study, because of the problems and difficulty that it entails. Conversely, the assembly forces for holes three and four are studied in order to check their low values.

Since only the reaction force in the assembly is studied and the locker is not a relevant feature for the model, a defeaturing process is performed to that area. Even more considering its complexity, convergence issues, the small flanges that conforms it and the contacts that could involve those critical areas.

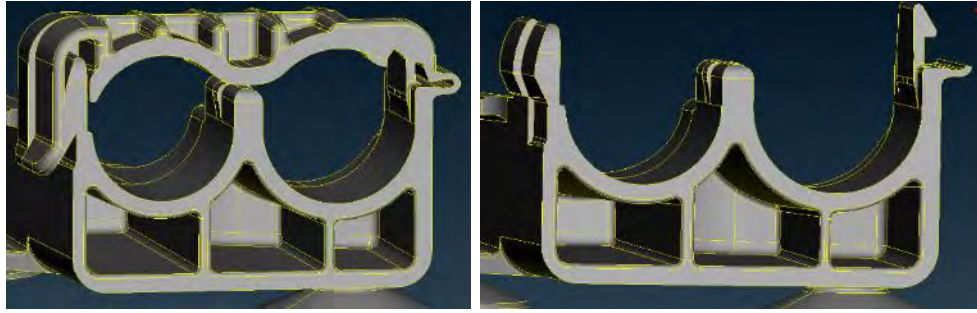


Figure 34. Locker defeaturing. left: before locker removal , right:after locker removal; Obtained from ANSA software.

Besides the locker removal, issues with non-convergence solutions appeared. It is necessary to improve the mesh in some critical areas, specifically in areas with surfaces involved in contacts, as shown in Figure 35.

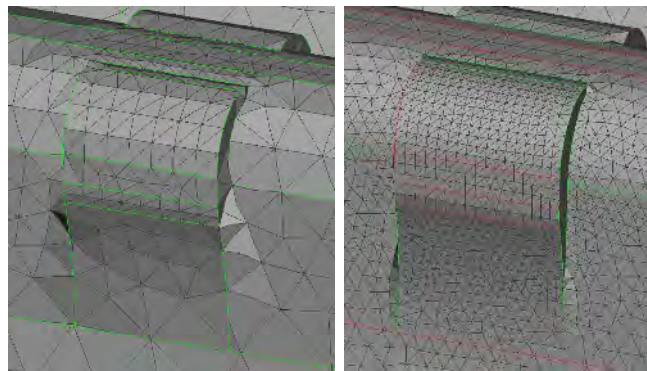


Figure 35. Mesh improvement in critical contact areas; Obtained from ANSA software.

4.3.3 Results

4.3.3.1 Reaction forces

During the explanation of this Chapter, the reader has been able to notice the performed changes between the Old and New design. During the timeline of the study the Old design has been developed in advance and many of the New design parameters seems to do not appear in the first one.

In order to obtain the best possible results for all models (Old design, Redesign, Assembly of hole 3 and Assembly of hole 4), the relevant parameters of the New design are also implemented in those models. This last improvement will allow obtaining accurate enough results for the upcoming correlation with the real test.

The assembly and pull-off forces obtained with the most suitable parameters are shown in the following Table 7 and Table 8.

Old design

Table 7. Old clip design results.

	RF [N]	Fulfil Requirements
Assembly 1	52	≈
Pull- off 1	155	✓
Assembly 2	45	✓
Pull- off 2	150	✓

New design

Table 8. New clip design results.

	RF [N]	Fulfil Requirements
Assembly 1	38	✓
Pull- off 1	280	✓
Assembly 2	55	✗
Pull- off 2	200	✓
Assembly 3	1.5	✓
Assembly 4	17	✓

4.3.3.2 Strains

The maximum principal strains obtained in the simulation are the one shown in Table 9 and Table 10.

Old design

Table 9. Old clip design results.

	Max. Strain[%]
Assembly 1	47.9
Pull- off 1	90
Assembly 2	47
Pull- off 2	84

New design

Table 10. New clip design results.

	Max. Strain[%]
Assembly 1	35.1
Pull- off 1	160
Assembly 2	54.8
Pull- off 2	135
Assembly 3	8.4
Assembly 4	4.6

4.3.4 Redesign

The section explains the process carried out to improve the design of the new design model, with the aim of achieving accepted values by the requirements.

The results of the new clip design show bad results in the assembly of the line into the hole 2. The force obtained is higher than the maximum value established in the specifications, thus, it has been decided to make small modifications in the design, in order to check if there is any possible solution to fix this design, or at least, to take into account the modifications for future clip designs.

The redesign is based on the theory that the non-symmetry of the clips is not desirable, due to strain concentrations and due to the difference in the deformations. In this case, the unsymmetrical axis is the one shown in Figure 36.

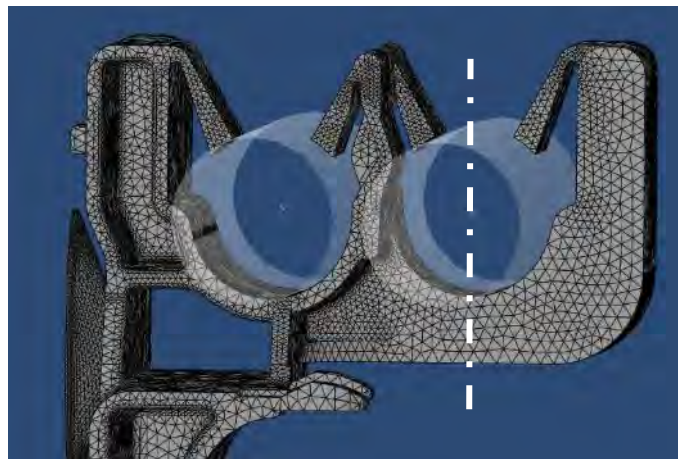


Figure 36. Non- symmetric axis in new clip design; Obtained from ANSA software.

It can be noticed that the left part of the hole 2 is so much wider than the right part. It produced a lack of equilibrium on the distribution of the forces applied by the fuel line or tube. During the insertion or removal of the tube, in the left part, the bending is much more pronounced, while the right part stays more rigid, and it causes high-undesired deformations on the left part. Focusing on the requirements, the assembly force is too high because it is difficult to deform and push the wide right part, so the redesign consists on removing some material on this part, in order to try to achieve a lower reaction force. One can think that this also will make decrease the pull-off force on that hole, but that force is high enough, so there is no need to worry in excess on that.

To make the hole, it has been used the actual shape in the inside of that part, as shown in Figure 37. The amount of material removed is not big, due to it is preferable to do it gradually.

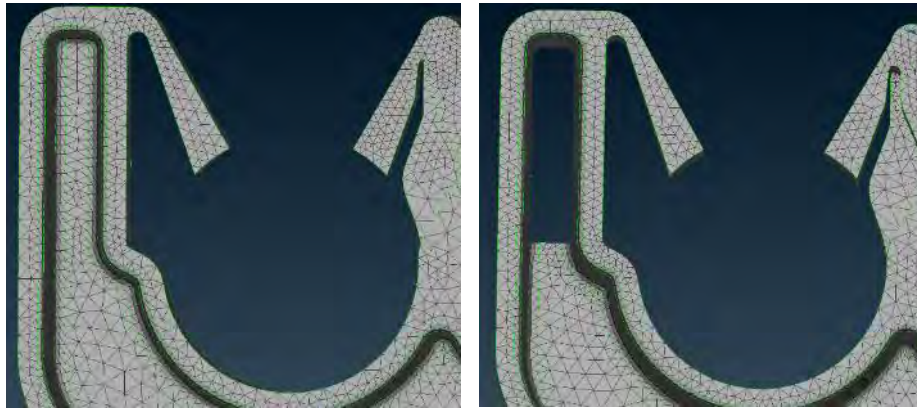


Figure 37. Re-design on hole 2. Left: original shape , right: hole 2 improved; Obtained from ANSA software.

Table 11. Comparison between new design and its redesign.

	New design RF [N]	Re-design RF [N]	Requirements
Assembly 2	55	52	<50
Pull- off 2	200	188	>35

The results show a predictable behaviour, a decrease in both reaction forces, although the reduction of the assembly force is not big enough. So, the next step could be to remove even more material from the right side of the clip until a proper force is obtained.

Table 12. Re-design of the new clip design results.

	Max. Strain[%]
Assembly 2	50 – 60
Pull- off 2	130 - 140

5 Real tests

The verification procedure for any clip based on the technical requirement is the real test. In this case, the plastic clips have a specific technical requirement, and some of the test acceptance criteria differ depending on small differences on the clips.

In this study, the clip is a standard model (not NVH or distance clips) and the clip has foot so it can be called a male clip. For this type of clip the testing conditions try to imitate the real conditions, and this is the reason of the several environments used for the test, as explained below.

5.1 Environmental conditions

The assembly is carried out at room temperature and room humidity, since those are the conditions of the real assembly line. The pull-off force is tested in order to make sure that the clips can hold the lines during the lifecycle of the vehicle, and that conditions mean variations in the humidity and the temperature. It is known, and later on verified, that the stiffness of the polyamide decreases with the humidity of the material, which means a worse behaviour, and therefore, the tubes will need less forces to get out of the clip. With this assumption, it is conceivable that the test should be performed in the worst situation, since a dry clip will always have a better behaviour. The pull-off test is carried out after humidify the clip at three different temperatures (-30 °C, RT and 80 °C).

It is important to highlight that the wet tests have been also carried out, although the TR does not require them. The wet tests are performed in order to make a deep and complete study about the polyamide behaviour, allowing to predict the material behaviour and to quantify the variation of the forces needed depending on the conditions.

All those requirements and different tests required for the assembly and pull-off force of the hoses into the clips can be summarized as the following:

Table 13. Technical requirements depending on test type.

	Conditions		Acceptance criteria
	Relative Humidity	Temperature	
Assembly	Room	RT	< 50 N
Pull- off	DRY 0 % humidity	-30 °C	> 35 N
		RT	
		80 °C	
	WET 95 % humidity	-30 °C	
		RT	
		80 °C	

5.2 Pre-test

Before starting with the test and preconditioning, it is necessary to organize all the clips and schedule all tests. The clips have to be numbered and the type of test that will be

carried on each one should be defined. Additionally, each hole should be numbered in order to facilitate their identification, as shown in Figure 38.

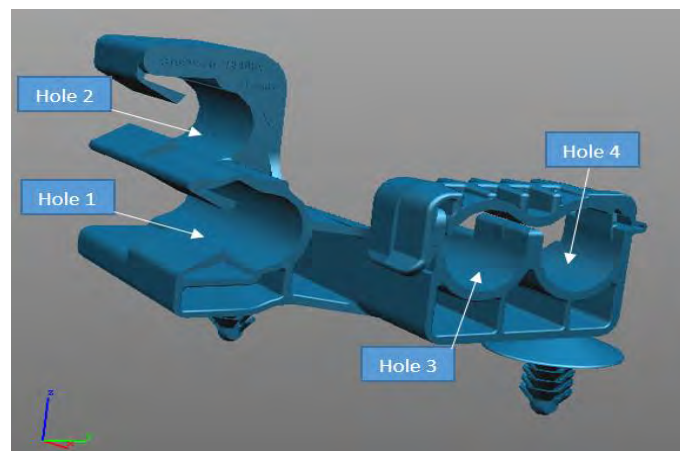


Figure 38. Hole's identification, New clip design; Obtained from Teamcenter software.

The clips are preconditioned depending on the test. On the dry tests, the clips remain during one week on a dry chamber in order to remove all the humidity and reach almost 0% of humidity. The roomRH clips test remain in a standard place with normal conditions around the laboratory, and the wet clips test require that the clip stay during 800 hours on a high humidity chamber (95 % humidity). It means that the wet test requires start with the preconditioning 2 months early, which is not desired. To reduce this time, the humidification process could be replaced by just immersing the samples on water during approximately 48 hours.

In order to get the most realistic test conditions, the clips are inserted on metal sheets. Those sheets have the same thickness as the parts used in the vehicle, and the holes made on it has the same diameter. The sheets also need two extra holes in order to attach it with a metal bar, which will help to locate the clip on the right position during the test, as shown in the Figure 39.



Figure 39. Metal sheet and fixation points.

Special care must be taken while making the holes on the sheets. The holes have to be done in a way that the clip is perfectly aligned with the machine, to obtain a uniform force all along the clip. Also, the holes made for attaching the sheet on the bar has to be on a right position, far enough to give space for the clip, but not too far in order to avoid the bending of the thin sheet. The Figure 40 shows what should be a good distance for making the holes (painted holes), since the bending of the metal sheet was realized during the tested.



Figure 40. Sample 1 with right hole's distance & sample 3 with wrong hole's distance.

On those clips that requires “special conditions” all the auxiliary tubes has to be inserted on the holes before the preconditioned, so that the tubes also reach the same conditions. The clip also needs to be already inserted on the metal sheets for the preconditioning.

5.3 Equipment

The tests are carried out on a tensile testing machine, and when the conditions are not RT and roomRH is needed a special chamber to acclimate the environment during the test, in order to keep the specific mechanical properties on the clips. The software used to save the results is TestExpert II, which is linked with the machine and it able to store all the time- force graphics in excel format.



Figure 41. Tensile machine and chamber at the back.

5.4 Testing steps and procedure

To obtain accurate results and a trustable range of values, each test needs at least five clip samples. The sequence of the tests is explained for new design model clip. If another type of clip is required, there is need to extrapolate those steps according to the number and type of holes.

Therefore, five different parts are need to be tested in each clip sample of the new design; hole 1, hole 2; hole 3 or hole 4 and the foot. It is important to develop those different tests in a specific order as below is explained.

The most critical values are the correspondent to the holes 1 and 2, since the holes 3 and 4 has a locker on them, and lockers have low assembly forces and high pull-off forces. For that reason, the assembly force is not going to be tested neither in hole 3 and hole 4. It is necessary to be careful with the adjacent holes, like 1 and 2, since the insertion or disinsertion in one of the holes could cause permanent strains that could affect the behaviour of the second test. In order to obtain accurate results, each sample will start with a different hole, so at least one can obtain some “virgin” values, both in assembly and pull-off tests. For testing the locker, there is need to alternate the tested hole for each sample, since once is tested the locker is not reusable for the other hole.

During the tests, both assembly and pull-off, an auxiliary tube has to be inserted on the non-tested hole, but it is important that this tube has been inserted through the longitudinal tube axis, avoiding make an extra insertion on the clip, which will cause undesired results.



Figure 42. Test of hole 3 with auxiliary tube in hole 4. New clip design.

The feet that hold the clip into other vehicle components is also tested for the pull-off test, although it is not simulated, so the results are not used in this report. To carry out the test a metal wire is placed all around the clip and is hooked into the steel pipe of the tensile machine.

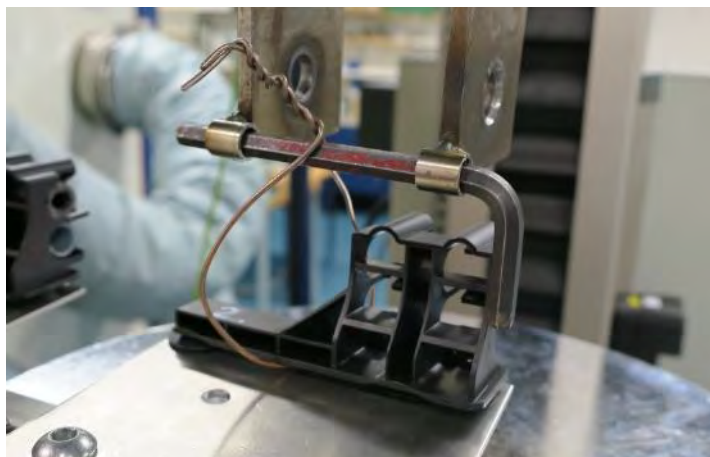


Figure 43. Foot test. Old clip design.

Both, the holes and the foot are tested using an Allen key or hex key, which diameter should be the higher that can fit into the tube, as shown in Figure 44.



Figure 44. Test of hole 2 with auxiliary tube in hole 1.

Summarizing, the test of the clip is carried out in the order explained in the following Table 14 for the pull-off and Table 15 for the assembly forces.

Table 14. Steps for pull-off force test.

Sample	Tested hole	Non- tested holes	Others
1	1	Insert auxiliary tube 2 laterally	Virgin sample
	2	Insert auxiliary tube 1 laterally	
	3	Insert auxiliary tube 4 laterally	
	Foot	Do not need auxiliary tubes	
2	2	Insert auxiliary tube 1 laterally	Virgin sample
	1	Insert auxiliary tube 2 laterally	
	3	Insert auxiliary tube 3 laterally	
	Foot	Do not need auxiliary tubes	
3	1	Insert auxiliary tube 2 laterally	Virgin sample
	2	Insert auxiliary tube 1 laterally	
	4	Insert auxiliary tube 4 laterally	
	Foot	Do not need auxiliary tubes	
4	2	Insert auxiliary tube 1 laterally	Virgin sample
	1	Insert auxiliary tube 2 laterally	
	4	Insert auxiliary tube 3 laterally	
	Foot	Do not need auxiliary tubes	
5	<ul style="list-style-type: none"> · If any of the previous tests presents a bad behaviour, or problems occurs while performing the test, that test should be repeated. · If not, the test to be develop is of free choice. 		

Table 15. Steps for assembly force test.

Sample	Tested hole	Non- tested holes	Others
1	1	Insert auxiliary tube 2 laterally	Virgin sample
	2	Auxiliary tube 1 already inserted	
2	2	Insert auxiliary tube 1 laterally	Virgin sample
	1	Auxiliary tube 2 already inserted	
3	1	Insert auxiliary tube 2 laterally	Virgin sample
	2	Auxiliary tube 1 already inserted	
4	2	Insert auxiliary tube 1 laterally	Virgin sample
	1	Auxiliary tube 2 already inserted	
5	<ul style="list-style-type: none"> · If any of the previous tests presents a bad behaviour, or problems occurs while performing the test, that test should be repeated. · If not, the test to be develop is of free choice. 		

5.5 Post- test

In order to get a complete test study and to be able to make a possible post-check, it is important to save all the useful information in an organized way. All the clips have to be marked with the number of the sample, and the graphs collected by the software have to be properly named with the number of the sample and the number of the hole. The general behaviour of the clips and any weird conduct should be reported during the performance of the tests. Also, all the tested clips should be stored in the same place with its proper identification using individual plastic boxes.

5.6 Test results

The followings Table 16 and Table 17, summarize all the data obtained from the testing. Not all the test results are collected on them, since there are too many values and could be confusing. The results that seem to be spurious and the results from test with some drawbacks are not consider in the solutions. For a better explanation, the Tables show the minimum force in the assembly test, the maximum force in the pull-off test, the average force calculated with the five samples in each of the test, and redundantly, the range of values obtained. When performing the verification of the results, it is important to take into account the worst possible case. That is the reason why the results of the minimum force for the assembly and the maximum force for the pull-off are not shown.

Naturally, all the results have been saved, so if there is any doubt during the verification, one can access to all the data and check all the results and comments.

Table 16. Old clip design test results.

Hole	Test type	Conditions (RH & Temp.)		F _{min} (N)	F _{max} (N)	Average F (N)	Interval (N)
1	Assembly	Room	RT	-	48.64	45.33	41.16 - 48.64
	Pull off	Room	RT	130.79	-	186.05	130.79 - 312.11
2	Assembly	Room	RT	-	47.97	46.77	44.67 - 47.97
	Pull off	Room	RT	111.97	-	136.86	111.97 - 152.96

Table 17. New clip design test results.

Hole	Test type	Conditions (RH & Temp.)		F _{min} (N)	F _{max} (N)	Average F (N)	Interval (N)
1	Assembly	Room	RT	-	50.66	44.11	38.21 - 50.66
	Pull-off	DRY 0 % RH	-30 °C	217.52	-	272.64	217.52 - 304.65
			RT	103.50	-	173.06	103.5 - 236.09
			+80 °C	100.11	-	102.28	100.11 - 122.17
		WET 95% RH	-30 °C	46.45	-	80.71	46.45 - 106.84
			RT	48.87	-	57.87	48.88 - 63.76
			+80°C	39.27	-	43.30	39.27 - 46.26
2	Assembly	Room	RT	-	80.81	69.66	52.88 - 50.66
	Pull-off	DRY 0% RH	-30 °C	210.58	-	232.30	210.59 - 265.01
			RT	159.28	-	188.50	159.28 - 214.45
			+80°C	90.28	-	94.16	90.28 - 111.42
		WET 95% RH	-30 °C	65.04	-	88.24	62.04 - 99.96
			RT	45.18	-	48.78	45.18 - 53.77
			+80°C	34.80	-	40.39	34.80 - 46.80

Having a look at all the results, and taking into account the technical requirements that needs to be fulfilled (maximum of 50 N for assembly force and minimum of 35 N for pull-off force), one can find that there are three different values that does not fulfil those specifications.

- Assembly force at hole 1. The maximum force reached is 50.66 N, and the maximum allowed is 50 N. Having a look at the range of results, one can see than only one sample has reached that value and that the minimum reached is so much lower, so it can be consider that the clip fulfil this requirement.
- Assembly force at hole 2. The maximum force reached is in this case is 80.81 N, a result totally unacceptable. Even the minimum force reached is higher than the maximum allowed, so this part should be reported and checked.
- Pull-off force at hole 2 in wet conditions at 80 °C. As happen with the assembly of the hole 1, the result is really close to fulfil the specifications, and the other samples works properly. So, again, this value could be consider as acceptable.

Thanks to the several test carried out in the pull-off test, it is possible to make a survey about the behaviour of the polyamide in different conditions. The Polyamide 66, or Nylon 66, is a crystalline thermoplastic commonly used in fabrics, military equipment or mechanical parts due to its properties such us toughness, strength and elasticity. It is known that its mechanical properties are very sensitive to changes in temperature and humidity, and the aim of this study is to see those changes and try to quantify them.

The followings Figure 45 and Figure 46 correspond to the test results of the pull-off forces in holes 1 and 2 respectively. They show six different curves, three for the wet

test and three for the dry test, and the three points in each curve represent the different temperatures.

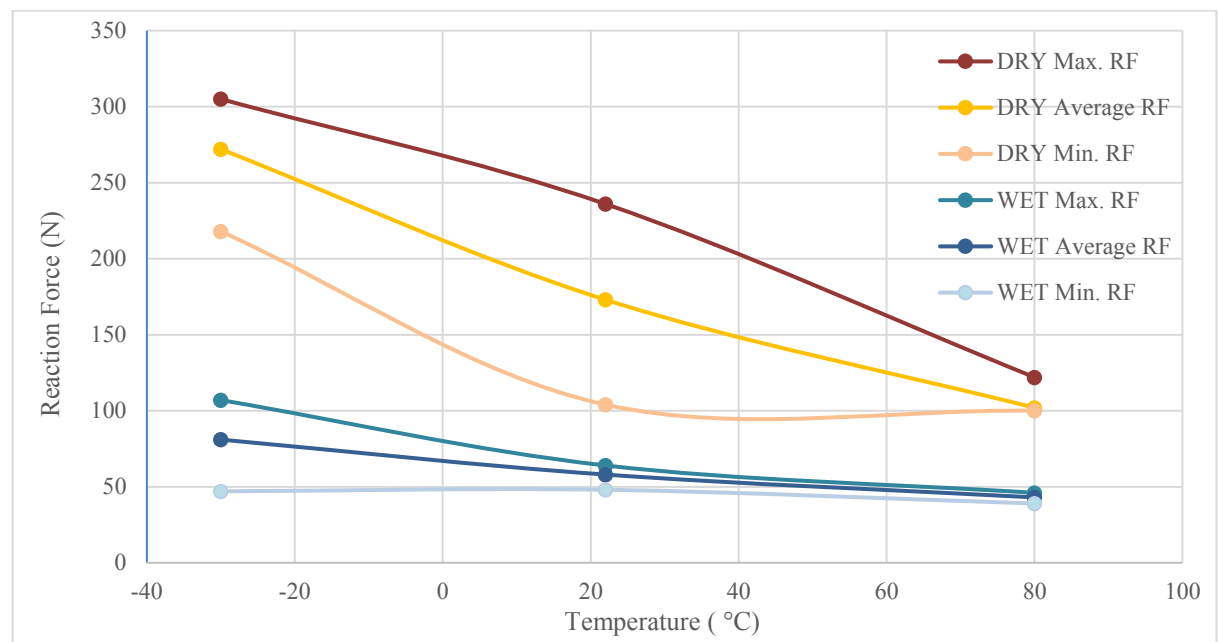


Figure 45. Pull-off force hole 1 test results.

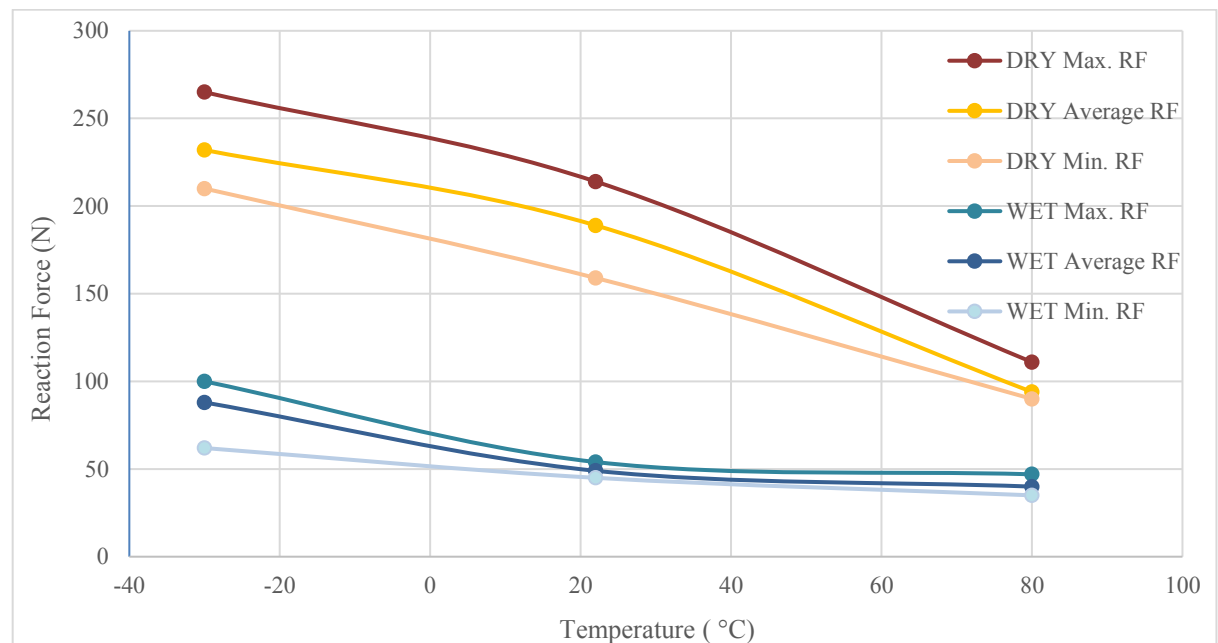


Figure 46. Pull-off force hole 2 test results.

Previously, it necessary to highlight that higher reaction forces means higher stiffness in the material, or what is the same, the Young Modulus increases.

Considering both graphs, there are two main statements that are unavoidable; the material requires more insertion force under drier conditions and under lower temperatures, which make sense. When the polyamide absorbs water, the molecules of

H₂O penetrate in the material, disordering the molecular structure and breaking some of the molecular bonds, so the movement inside the material is easier, which means that the stiffness decreases. Something similar happens with the temperature, but in a lesser extent. Lower temperatures tends to make stronger molecular bonds, making the displacement of the structure more difficult, and increase the stiffness of the material. Summarizing, regarding the pull-off forces the best environmental conditions are dryness and coldness, so that the fuel lines needs high forces to be removed from the clip, which is preferable due to security reasons. On contrary, is preferable to carry out the assembly in a room with higher humidity and temperature, so that the force needed to insert the fuel line into the clip is lower.

6 Results correlation

Once obtained all the results from the tests and once all the simulations have been completed, it is necessary to correlate and compare both results in order to check if the behaviour of the simulations is good enough, and if using exclusively a FE analysis would be a reliable verification method for at least the early stages of a design.

Naturally, several correlations have been done during the performance of the project. One of the most important task to carry out is to adjust the several parameters that involve the FE software in order to approach the simulation to the reality, and that parameters need to be changed by comparing the different simulation results to the real test results iteratively.

Moreover, in this section, only the final results achieved are shown, for both, old clip design and new clip design. It is important that at least two different types of clip are studied so that two different perspectives can be considered, since different shapes can present different behaviours and the aim of the project is to become with a suitable guideline simulation that works for different type of clips.

6.1 Considerations

First, there is need to consider any unavoidable difference between both models, and take in mind how those differences can affect the model for the correlation.

- The fixation point of the clips is so much restrictive in the simulation. When a boundary condition of fixation is applied on the model, the user is giving 0 mm of displacement on some nodes, which means that those nodes are not moving at all. However, on the real test the fixed feet tend to deform and move from its initial position when the force applied is high. This will cause lower bending of the clip in the simulation model.
- On the FE software, the properties of the polyamide PA66 are defined in conditions of RT and 50 % of RH. This is completely theoretical and could be some differences with the real behaviour of the material. On the other hand, the real test requires 5 samples for each conditions, but performing each test takes some time, and during this period the properties could change.
- Therefore, on the FE software, the properties of the material are defined for a specific type of polyamide 66, the Scanamid 66_A331E, and the specific type of polyamide 66 in the reality is unknown. Anyway, the mechanical properties would not differ so much, for the rest of the study, both materials are consider as the same.
- Small changes in some parameters can affect one simulation model and cannot affect other, so there could be small mismatches between models.
- It is not possible to fix each sample in the same position during the test, and any difference on the distance or alignment can change the test results. That is why there are 5 samples in each test, and during the correlation three different values are considered; maximum reaction force, minimum reaction force and the average value.

6.2 Old design model

As mentioned before, the old clip is tested in conditions of RT and room RH and the simulation has RT and 50 % of relative humidity. The humidity conditions of the lab could varies a bit, but it has been measured with a value of 28 % relative humidity, or 28RH. Taking into account the previous polyamide behaviour study, the material presents higher stiffness in drier conditions, so the forces needed to assembly or disassembly the line into the clip should be higher. That means that the results obtained on the test should give higher values than the simulation values. The reality, shown on Table 18 is that some of the results are higher, and other lower, but all of them has similar and close results, so there is one appointment that could be establish. The conditions of 28RH in the real test and 50RH in simulation can be compared and seems to correspond.

Table 18. Correlation between test and simulation results for assembly and pull-of forces. Old clip design.

	Test Results (RT + 28RH) [N]			Simulation Results (RT + 50RH) [N]
	Min. Value	Max. Value	Average Value	
Assembly 1	41.16 N	48.64 N	45.33 N	52 N
Pull- off 1	130.79 N	231.11 N	186.05 N	155 N
Assembly 2	44.67 N	47.97 N	46.77 N	45 N
Pull- off 2	111.97 N	152.96 N	136.86 N	150 N

The results present a really good behaviour in the simulation, even being the pull-off forces quite lower or higher. Obtain the exact same results is almost impossible, so obtaining those results, that are close, make sense and are proportional with each others, it is a really good achievement.

In order to quantify those differences, the relative error between the simulation and the real test is calculated. Consequently, the percentage of correlation between them is also achieved. It is remarkable that those values are calculated with the average value of the five samples of the real test, using equation (6.1) and equation (6.2). The results are shown on Table 19.

$$\begin{aligned}
 \text{Relative error (\%)} &= \frac{| \text{Real value} - \text{Exprimental value} |}{\text{Real value}} * 100 \\
 &= \frac{| \text{Average of real test} - \text{Simulation} |}{\text{Average of real test}} * 100 \quad (6.1)
 \end{aligned}$$

$$\text{Correlation (\%)} = 100 - \text{Relative error} \quad (6.2)$$

Table 19. Percentage of relative error and simulation.

	Relative error (%)	Correlation (%)
Assembly 1	15 %	85 %
Pull- off 1	16 %	84 %
Assembly 2	4.7 %	96.3 %
Pull- off 2	9.6 %	90.4 %

The percentage values of the correlation are high, around 90%. With those results, it can be assumed that the simulation is a good tool during the early-stages of the design of the clip, and it could even works as an accurate verification method.

6.3 New design model

It is needed to highlight that the new clip design is more complex in terms of simulation. The non- symmetrical features of the clip could add some difficulties on the model, and this is the reason why higher variations can appear when correlating this new clip values. Moreover, the correlation of the values of the pull-off test in the new clip design are more complex, since those test have been carried out on different ambient conditions (varying temperatures and range of relative humidity). Due to it, the pull-off forces for the new clip design are represented on a graph, allowing a better and visual understanding of the correlation. The assembly force correlation is shown in Table 20, in the same way as the data of the old design. The environmental properties are the same as in the old clip design, so it is supposed that the results should be similar, but, again, some differences could appear due to the variation in the humidity of the laboratory.

Table 20. Correlation between test and simulation results for assembly forces. New clip design

	Test Results (RT + 28RH) [N]			Simulation Results (RT + 50RH) [N]
	Min. Value	Max. Value	Average Value	
Assembly 1	38.21 N	50.66 N	44.1 N	38 N
Pull- off 1	-	-	-	280 N
Assembly 2	52.88 N	80.81 N	69.66 N	55 N
Pull- off 2	-	-	-	200 N

In this case, both assembly forces are quite lower in the simulation model, but again they are proportional and express close values, so the simulation can be consider as good enough.

Again, Table 21 shows the percentage of the values and the percentage of the accuracy of the simulation.

Table 21. Percentage of relative error and correlation.

	Relative error (%)	Correlation (%)
Assembly 1	14 %	86 %
Pull- off 1	-	-
Assembly 2	21 %	79 %
Pull- off 2	-	-

In that case, the percentage values of the correlation for the assembly forces are a bit lower, but still good. Therefore, it can be assumed that the correlation is good enough for early stages, but not recommendable as final verification method.

The correlation of the pull-off forces of the new design are more complex. As stated above, the pull-off tests has been developed in different conditions; Three test in dry conditions (0 to 5 % of RH), and three test in wet conditions (95% RH). The three temperatures are -30°C, RT and 80°C. It would be interesting to have also normal conditions test (RT and room RH), so that the pull-off force could be correlated in the same way as the assemblies. However, this clip needed to be tested according to the TR, since the clip is currently in production and it is actually being used in the vehicle, so the limitations of time, clip samples, and availability of the laboratory equipment did not allow the realization of the test. Anyway, the entire test with variations on those conditions is interesting to see the behaviour of the polyamide.

The followings Figure 47 and Figure 48 correspond to the pull-off forces in holes 1 and 2 respectively. They show six different curves, three for the wet test and three for the dry test. As in the other correlations, the maximum, minimum and average values of the test are represented, providing more information.

Therefore, to carry out the correlation, it is supposed that the values of the simulation should be located on 22 °C in X axis. About the humidity is established before that 50 RH in simulation seems to behave like the normal conditions of 28RH in the real test, so the value is being looked for should be closer to the dry tests than to the wet tests. The reality is that, in both graphs, the value obtained in the simulation is higher than the ones obtained during the test, and the values are even higher than the dry test values. That means that the simulation model is showing a clip more stiffness than expected, so the tube needs more force to deform the wings in order to get out.

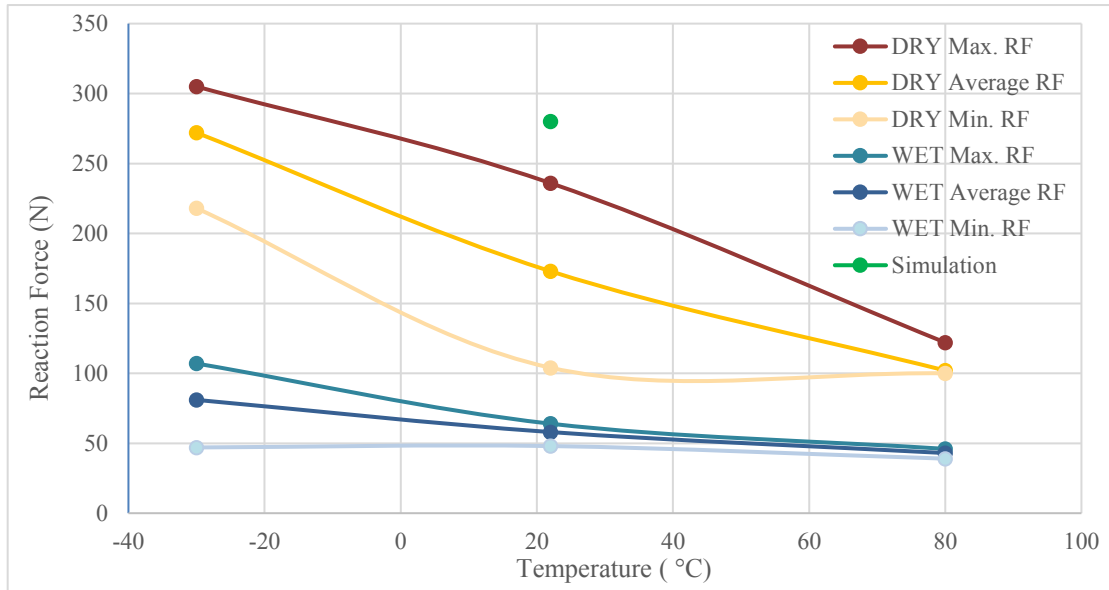


Figure 47. Correlation between test and simulation results for pull-off force at hole 1. New clip design.

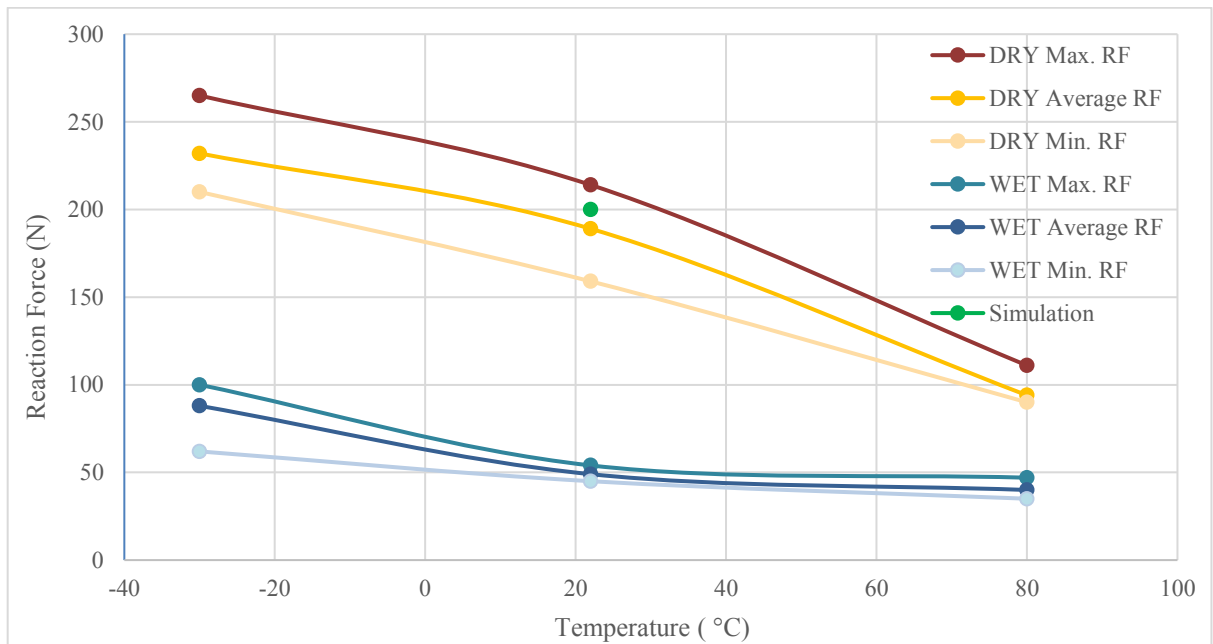


Figure 48. Correlation between test and simulation results for pull-off force at hole 2. New clip design

One can think that one of the reason is that the friction coefficient is high in both holes, but the correlation of the old design has been made with the same coefficients and it shows a good behaviour. The reason could be the complexity of the model, as explained before, since the clip is not symmetric at all, and that causes high deformations and displacements, which is not desirable during the FE analysis. Furthermore, there is need to emphasize that the pull-off force is always more difficult to carry out, since the wings suffer high variation in their shape.

For the purpose of this project, achieve a good simulation tool that allows making early-stages verifications, the correlation of the reaction forces is the most important step. However, it is also essential to compare and correlate the general behaviour of the clip during the whole process of insertion or removal. That is, check if the movement of the wings and of the whole clip in general are similar, in accordance with the strains achieved in the simulation. With the help of videos and pictures taken from the real test it is possible to roughly compare the behaviour of real test with the behaviour of the deformed shape in the simulation.

Table 22. Comparison real test vs simulation. Old design

	Real test	Simulation	Strain [%]
Assembly 1	Normal behaviour + Bending	Normal behaviour + Bending	48
Assembly 2			47
Pull- off 1	Right wing up (broken)	Both wings up (more right one)	90
Pull- off 2	Right wing broken	Right wing up	84

In the case of the Old design, as explained above, the study achieve a good correlation just comparing the resulting reaction forces, but even more could be validated once compared this last step, in Table 22. The simulation almost follows the same behaviour as the real test.

Significantly, both assembly and pull-off models have the same behaviour, below in the Figure 49 is shown the visual comparison of one of them.

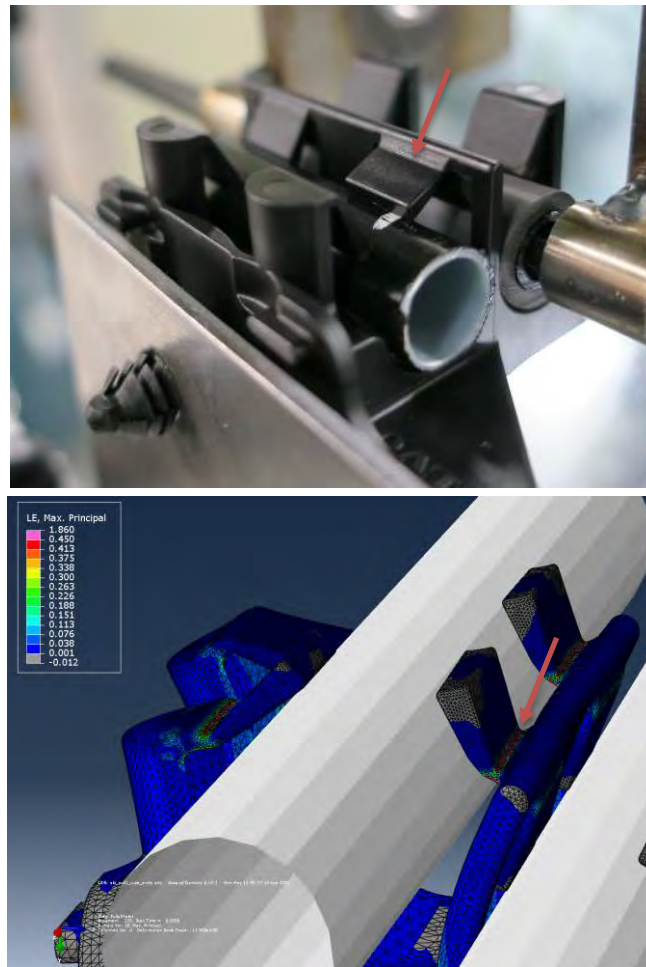


Figure 49. Visual comparison for old design pull-off; Up: real test, down: simulation.

Is possible to notice that the critical strain area in the simulation is the same area where the wing breaks in the real test.

Table 23. Comparison real test vs simulation. New design

	Real test	Simulation	Strain [%]
Assembly 1	No information available	Normal behaviour + Bending	35
Assembly 2			55
Pull- off 1	Dry: both wings up	Both wings up (more left one)	<160
	Humidity: left wing up		
Pull- off 2	Dry: right wing broken	Left wing up	<135
	Humidity: right wing up		

The correlation of the new design is more complex and the numerical results achieved differ more than the old design. Despite this, the visual comparison yields good results. The assembly tests are not relevant due to no visual information is available from the real test. The pull-off tests, as explained above, are not available in 28 RH conditions, but some similarities can be noticed between them and the simulation.

In the Figure 50, the pull-off forces are shown, in order to clarify the Table 23. Again is possible to notice a clear strain concentration in the simulation around the breaking area of the real test.

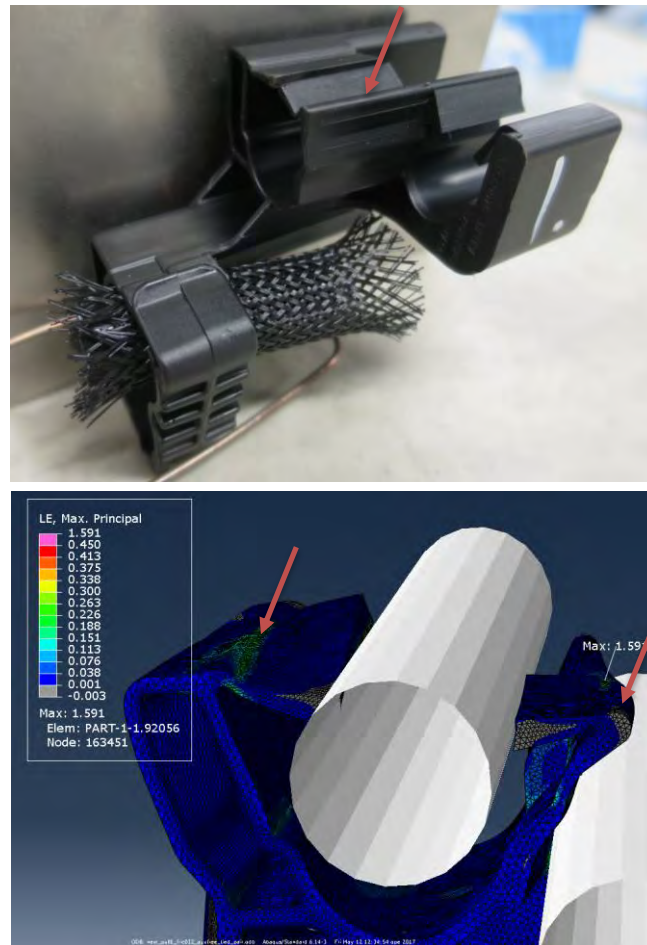


Figure 50. Visual comparison for new design pull-off. Up: real test, down: simulation

7 Recommendations for upcoming designs

The following recommendations are obtained based on the deep study of two clip models, the new design and the old design, and also in the results of the re-design carried out in the second one. These tips are focused on improving the design of those clips or similar ones, but naturally, they can be extrapolated and applied to other type of clips.

7.1 Symmetry

In general terms the behaviour of symmetric, or at least very similar wings is better, in the Figure 51 one can see the lack of symmetry trough the transverse axis. This produce strain concentrations in the linking point of both unsymmetrical parts. Therefore, it is important to follow certain symmetry in the designs, in that way the strains and efforts will be distributed equally between both wings.

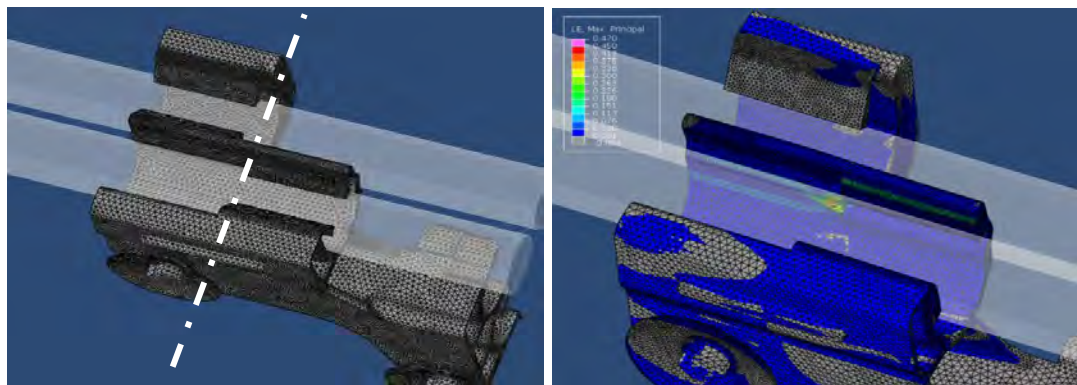


Figure 51. Asymmetry in transverse axis; Obtained from Abaqus viewer software.

In the following Figure 52, the reader can observe another axis of symmetry, in this case the vertical one. This is a special type of symmetry, in order to take in mind the shape in both sides of the axis. The right part of the axis is thicker, producing a higher deformation/bending in the left and thinner part of the axis. It is important for upcoming designs to try to achieve the same amount of material in both parts of the axis. The reader has been able to observe the benefits of a better-balanced amount of material in the Chapter 4, Section 4.3.4 Redesign, previously explained in this report.

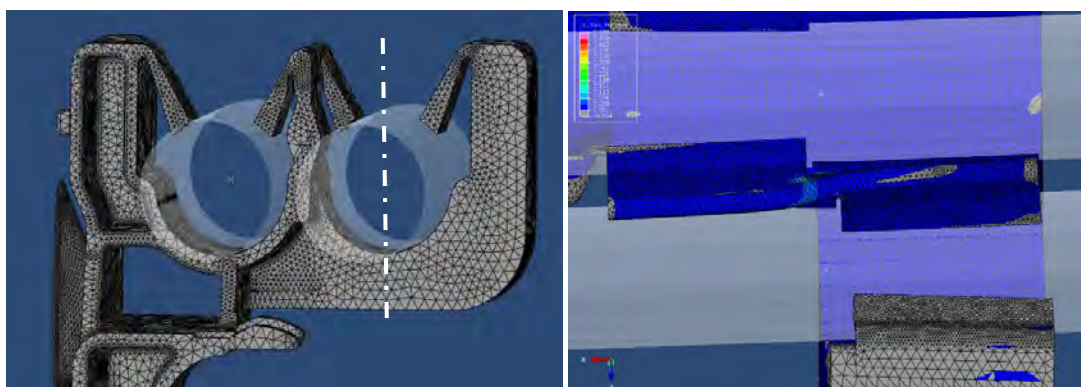


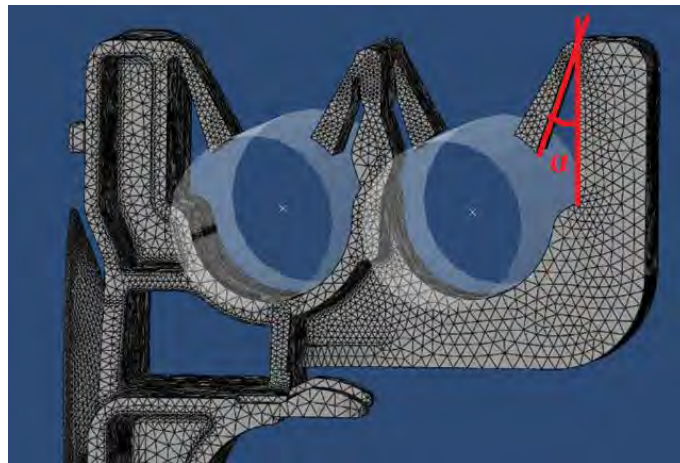
Figure 52. Asymmetry in transverse axis; Obtained from Abaqus viewer software.

7.2 Thickness of the wings

The most critical part of the wing is the upper area, that is, the linking part between the wing and the wall, since that part concentrates all the strains, and consequently, it is the weakest point, thus an excessively narrow wing is not recommended.

A wide wing will cause higher forces in both assembly and pull-off, so in case the designer needs to decrease those forces without making a narrow wing, there are some others alternatives.

- Decrease the angle α formed with the wing and the wall. The assembly force will be reduced –which is desirable- and the pull-off is more unpredictable. In case that the wings tends to go up in the pull-off, the force will be higher. In case the wings tends to stay down during the pull-off, the force will be lower.



*Figure 53. Angle formed by the wing and the wall;
Obtained from Abaqus viewer software.*

- Decrease the length of the wings. Both forces will also decrease, but maybe in a more discrete way.
- Reduce the stiffness on the structure of the wall, or other areas that hold the wings. It will made that the whole clip can bend more during the insertion of removal of the tube, while the deformation of the wings decrease.

7.3 Corners, edges & shapes

It is recommendable to make smooth surfaces avoiding sharp edges or rough changes in the shape, since those areas are the most critical ones in terms of unexpected and undesirable strains and efforts concentrations.

It is also recommendable to implement round corners in places where are going to be high concentrations of strains, and as higher the radius of the round is, the less strains will appear.

Summarizing, as simpler, more symmetric, less complicated and without excessively thin parts, the better will be its behaviour.

8 Conclusions

Once done all the simulations, real test, correlation, and all the comparison and studies, some conclusions about the general behaviour are reached. The following conclusions are focused on the relation between the simulation and the real test, more than comparing the simulation with the real behaviour of the clip in the car.

The most important thing to highlight is that the process established on the Chapter 9. Technical recommendation for clip simulation can be consider as a good verification method, at least, for early stages of the clip. Even though some correlations shows reaction forces quite different from the real values, in all cases the simulations replicate really well the behaviour of the clip. For for making an early verification during the design process of the clip, this may be enough.

It is necessary to emphasize that the real test method is not going to be replaced by the simulation method, or at least, not yet. Both verification methods will be complementary; the simulation will help to select the best CAD design model possible by comparing the approximate results, and the real test will be the reliable and final verification method to check the exact forces of the tests for fulfil the requirements. The benefits of having this simulation tool are obvious. Thanks to the simulation pre-check, any clip with bad behaviour can be detected and removed during the realization of the design, avoiding the unnecessary waste of money and time due to sending that clip to production and then, making the real test.

In addition, it is important to mention the general reasons why sometimes the correlation could not be as accurate as expected, since it is essential to understand all the possible issues behind the simulation, even if they cannot be fixed.

- The most important reason could be the difference between the real test scenario and the simulated conditions. The way of fixing the foot into the metal sheet is a simplified approximation of the reality; the tube that imitates the fuel line is a rigid body with the friction coefficient of the plastic real material, while in the reality it is a steel (rigid) pipe covered with the material; during the simulation, the insertion and removal of the tube is perfectly aligned, while in the real test, as in the reality, the alignment could not be that good.
- Regarding the real test, the reader can see the broad range of achieved results. Those big differences on the same test between different samples could be caused due to different alignments, different forces applied on the metal sheet that hold the clip, or small differences on the tubes used. Summarizing, the worker action on the test is critical, that is why the test should be done in a proper and systematic way. The result is a wide range of values obtained from the real tests, which makes the correlation even more complex and less accuracy.
- Other cause is related with the material, due to two main reasons. The first one is related with the differences of the environmental conditions that the material is exposed to. The simulation software contains a package with the different mechanical properties of the polyamide, which are standard. The problem is that

while the simulation remains with the same material conditions, the test conditions could change a bit, since it is impossible to achieve the same exact conditions requested in all the clips during the whole test (50RH, 0RH, 95 RH or -30°C, RT, +80°C). In addition, the normal or standard conditions on the laboratory can differ a bit. Again, it is obtained a big range of real test results.

- The other issue found out during this project regarding the material is that there are different types of polyamides 66 with small variations on their conditions. The one used in the simulation, is Scanamid 66_A331E, while the type of polyamide 66 used in the reality is unknown (at least it could not be provided). Therefore, assuming that the polyamide used in the reality is not the same as the polyamide on the material library, the mechanical properties can differ a bit, and consequently, the solutions.
- There are also other factors related to the FE simulation, more technical, unpredictable and tricky to solve, which affect the solutions. Those parameters governs the execution of the equation solving process, such as the type of analysis, values related with the step and the time and so on. In addition, all the BC, contacts, and loads applied increase the complexity of the model, adding more equations to be solved. These could all lead to an increment in the possibilities of having convergence issues and not reaching to the final solution. Each model is very different, and also its equations, so in case there is need to fix the convergence issues, each model will need a specific and individualized parameter setting. This could involve small differences in the model and thus, in the solution. It was one of the biggest challenges in the project, since the comparisons of the models should be made with the exact same parameters, and each time there is a different convergence problem, all the models had to be reset again to make them as much similar as they could.

Besides, as conclusion of the study, some other improvements have been developed.

- The methodology of the real test has been defined in an organized way with more information, since the current technical requirement can be ambiguous and has lack of information. All the different real test have to be carried out in the exactly same way in order to obtain results than can be compared.
- The tips and hints provided to the designers allows a better clip design, avoiding to repeat the same failures of previous designs.
- More specific, the new clip design does not fulfil some of the requirements. With the deeply studied performed on it, it was possible to provide some solutions and to give a better understanding of its behaviour.

In summary, the simulation method obtained from this study is a good enough verification method, but only for the early stages. There is also need to take in mind and understand all the factors that involve the simulation and could affect it, in order to have a deep understanding of the results achieved. Moreover, it has been noticed that the clip's simulation is really complex, requiring time and different tries, and each model has each own difficulties and challenges.

9 Technical recommendations for clip simulation

These recommendations can serve as a guide when solving problems or questions that arise during the development or simulation of the clip model, but one should understand that without a basic knowledge of the Finite Element Method, ANSA and ABAQUS, the understanding of these recommendations could be difficult.

It is strongly recommended to read and take into consideration during the understanding of this report the following documents:

- Ansa User's guide
- Abaqus User's guide
- Getting Started with Abaqus: Interactive Edition
- Getting Started with Abaqus: Keywords Edition

During the following parameters setting up, some features are probably referenced to the Chapter 4. Clip simulation, since the following steps are based on the iterations carried out with all the simulations.

The following Figure 54 show an overview of the user's interface in Ansa, in order to clarify the terminology that the reader will find in this chapter.

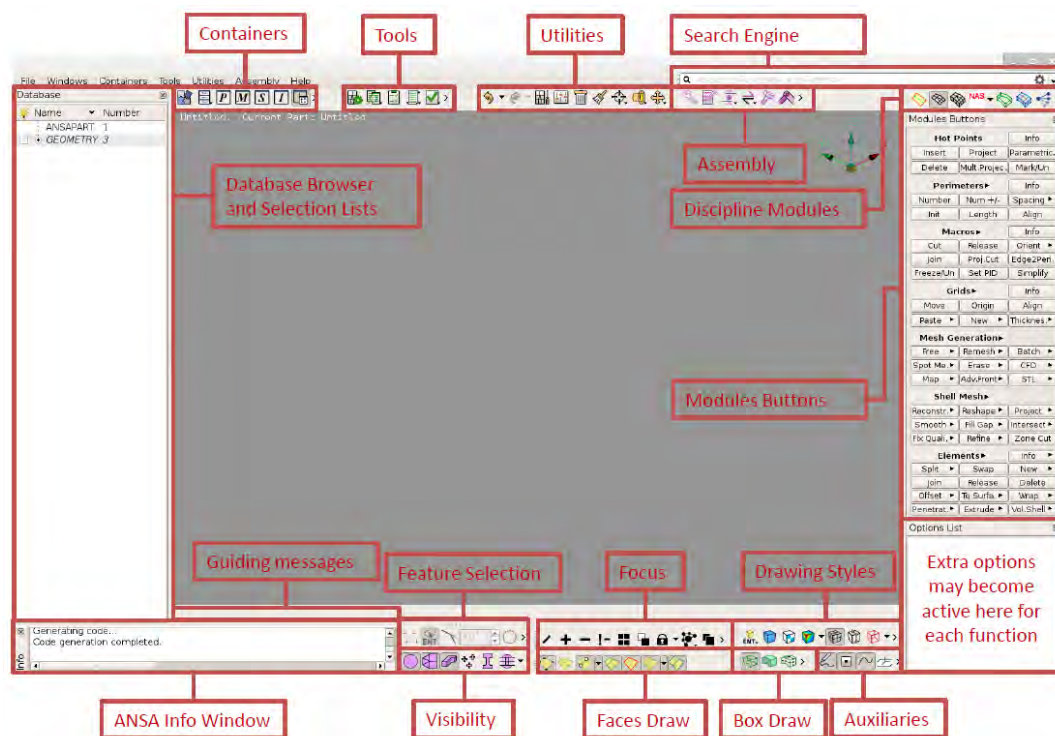


Figure 54. User interface of Ansa software. Obtained from: *Getting started Ansa.pdf*

In order to facilitate the object visualization, the user can situate the “camera” in pre-set views, with the F5 to F10 keys.

During the entire simulation process is important to label each function, such as sets, contacts steps, etc. in a proper way, in order to obtain a clean and easy to check/understand result.

9.1 Settings

Tolerance settings

Define tolerances

Windows → settings → tolerances:

- Tolerance mode = fine

Recommendation: Tolerances must be less than the minimum element length of the mesh. See Chapter 4, Section 4.3.2.1 Topology and resolution.

Resolution

Define resolution

Windows → settings → resolution:

- Curves = 10
- Perimeter length = 0.1
- Distortion distance = 20%
- Distortion angle = 35

Recommendation: One should take into account that the perimeter length of CONS resolution should be at least the same as the target element length of the mesh or finer, allowing a proper visualisation of the “real” mesh. See Chapter 4, Section 4.3.2.1 Topology and resolution.

9.2 Import geometry

Switch to the Abaqus solver deck module

- File → Open → *namefile.CATpart*

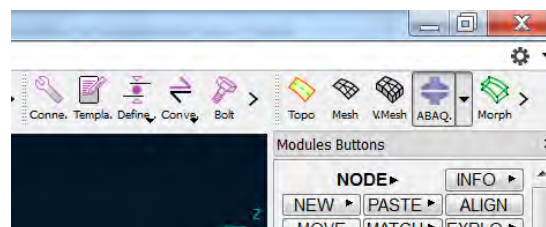


Figure 55. Discipline modules; Obtained from ANSA software.

Recommendation: It is possible to create the geometry with ANSA, since even not being a CAD software, it allows to make some design modifications with no high complexity. The user will find the respective tools in the Topo module. See Figure 55.

9.3 Check & clean up

Check the imported geometry

- Checks tool → geometry → Auto. Fix (See Chapter 4, Section 4.3.2.3 Check and clean up)

Recommendation: When some needle face appear, it is necessary to check it carefully, avoiding an automatic fixation. The automatic tool would delete the face, leaving a gap in the geometry model. Example explained below:

Close the square perimeter

- Topo module → faces → cut

Delete the problematic faces

- Faces → delete

Create a new surface (see Figure 56. middle)

- Surfaces → Coons

Recommendation: One can consider in the mesh module join those macros in order to obtain a smoother mesh.

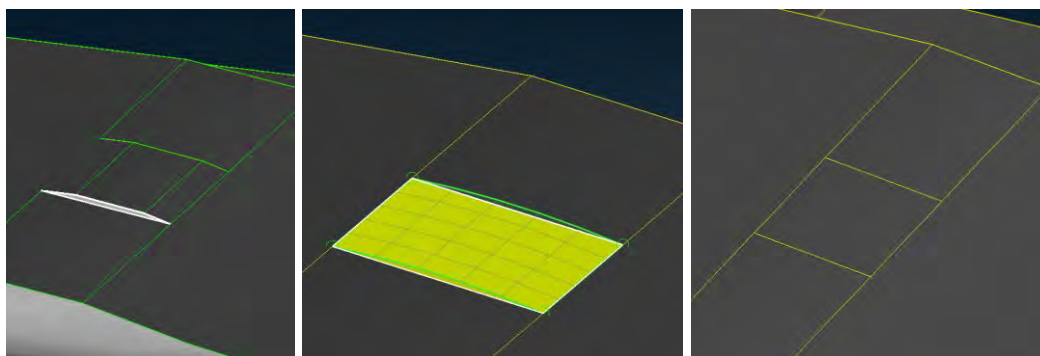


Figure 56. Fixing of needle faces. left: needle face , middle: surface creation , right: after clean up; Obtained from ANSA software.

Recommendation: Different models may have different geometry issues, the user shall fix them avoiding complicate the model excessively.

9.4 Defeaturing

Recommendation: As explained in the Chapter 4, Section 4.3.2.4 Defeaturing, non-necessary features must be deleted. Explained below the user find how to remove the logos, clip's feet, and locker of holes three and four, since less features means less mesh and less computational time, and some convergence issues can be avoid.

Logos

Remove desired logos

- Topo → Rm.Logos → per face

Recommendation: There are features with really small size and sharp edges. The logos can be easily removed using the previous tool and selecting the face where the logo is situated, for what is necessary first to measure the logo height.

Feet

Recommendation: One should take into account the way of fixing the clip to the component and decide if is important to keep the geometry of the feet, those can produce several mesh quality issues (in case that the mesh in those areas is not fine enough).

Deleting undesired faces

- Topo → Delete

Create a new plane in order to be able to draw on it

- Auxiliaries → Working planes → New

Situate the view looking into the bottom of the model

Draw a circle in the active work plane

- Curves → Circle

Create a surface from that circle and check the orientation

- Surface → Coons. Faces → Orient

Intersect + trim the circular surface created and the feet

- Faces → Intersect

Recommendation: In case a wrong trimming is made, one shall consider create a surface with Surface → Coons

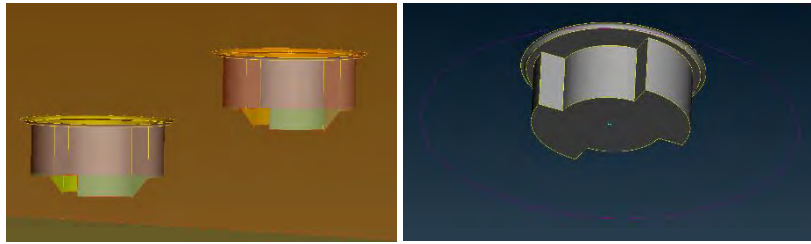


Figure 57. Feet defeaturing; Obtained from ANSA software.

Locker

Remove undesired lockers

- Topo → Faces → Cut
- Surface → Coons

Recommendation: One can easily trim the not desired part, using the previous tool between hot points (nodes on CONS representing their end points), for then deleting some faces and create the necessary surfaces.

Recommendation: The user must take into account that all the curves on surface that appears in the model -in the TOPO module- shall remain in yellow, meaning that are connecting two surfaces. If one can find a red curve, the face which belong to shall be removed and created again (by surface → coons)

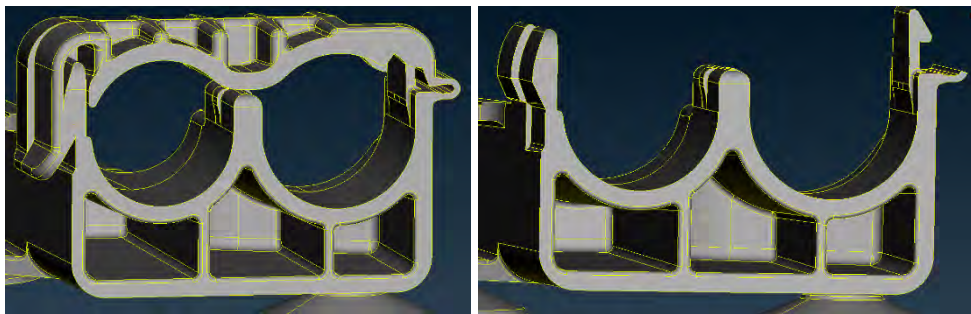


Figure 58. Locker defeaturing; Obtained from ANSA software.

9.5 Creating tubes

Recommendation: As it is explained in the Chapter 4, Section 4.3.2.8 Tube, the tubes shall be implemented as analytical rigid tubes. The reader should be aware that is not possible to implement directly an analytical rigid surface, it is necessary to create first the analytical surface and then assign it to a rigid body. Beware of the reference node, shall be the same for both features.

It is much simpler to create just one tube for then making a copy-paste obtain the auxiliary one.

Recommendation: The user shall create a 3D line with two points in its ends, in order to represent the tube axis (points *a* and *b*). The analytical surface is defined by the *profile*, where the user defines the radius and longitude of the tube. The resolution is defined in order to obtain the finest tube possible.

Create a new plane in order to be able to draw the tube axis on it

- Topo → Auxiliaries → Working planes → New

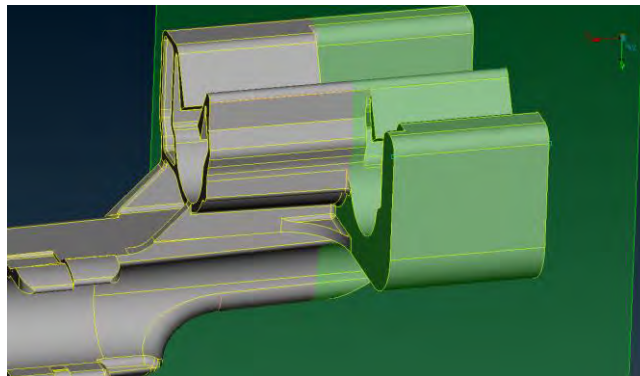


Figure 59. Workplane; Obtained from ANSA software.

Create the line with 2 points (a and b) in the working plane

- Curves → New → Pick
- Points → New → In line (select the created line) → Number of points = 2

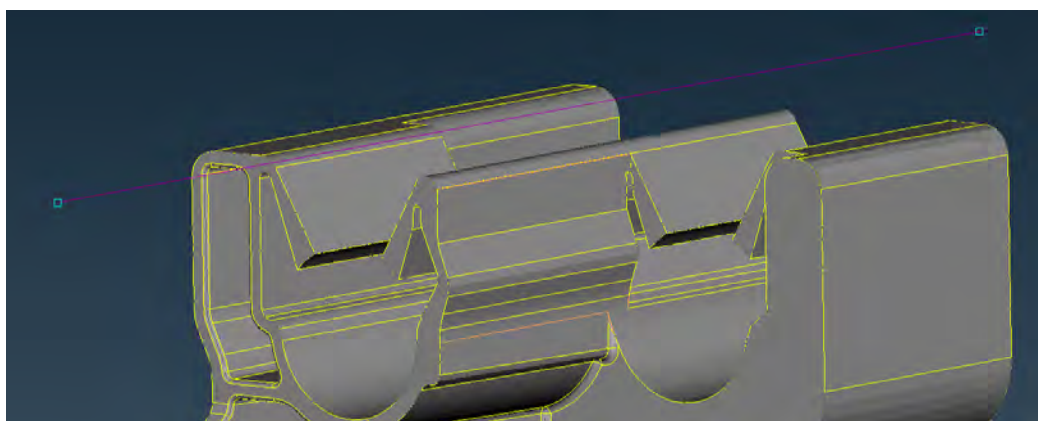


Figure 60. Axis of the tube; Obtained from ANSA software.

Recommendation: In order to be able to visualize the working planes and the auxiliary features the user must be aware of keep activated the remarked button in Figure 61.

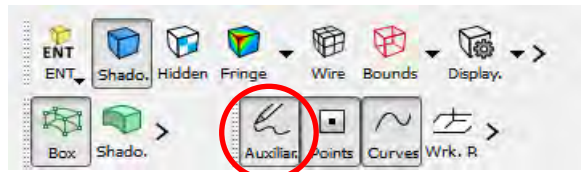


Figure 61. Axuliaries buttom; Obtained from ANSA software.

Rigid body

Create a rigid body

- Abaqus deck module → Auxiliaries → R.Body → New → Analytical surface

Analytical surface

Create analytical surface in the appearing window from the rigid body.

- Analytical surface → New:
 - Surface type = surface
 - Resolution = extra fine
 - Fillet radius = empty
 - $(X_a, Y_a, Z_a) = F1$ and click on the point “a” of the line
 - $(X_b, Y_b, Z_b) = F1$ and click on the point “b” of the line
 - Profile = Line, X_s = tube radii, Y_s = tube longitude, X_e = tube radii, $Y_e = 0$ (see Figure 62.)

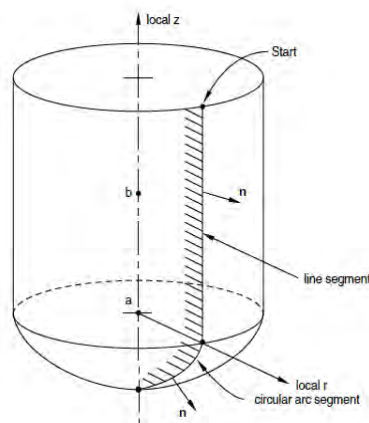


Figure 62. Analytical surface implementation in Abaqus software; obtained from: Abaqus interactive edition.pdf

Recommendation: As it is explained in the Figure 62, the surface normal direction depends on the order of selecting the parameters in the profile. The user must be aware

that the yellow part has to remain inside the tube, meaning that the surface normal is pointing outwards.

Recommendation: Once the analytical rigid surface is created, the user shall situate it in the desired position. Then is highly recommended to duplicate it and situate it in the desired auxiliary tube position:

- Transform utility → move → entities → select the tube
Transform utility → copy → entities → select the tube

Recommendation: When situating the tube, the user should notice the necessary displacement to insert the tube in the hole, in this case it is 13 mm in the global Y direction, for them applied this displacement in the boundary conditions.

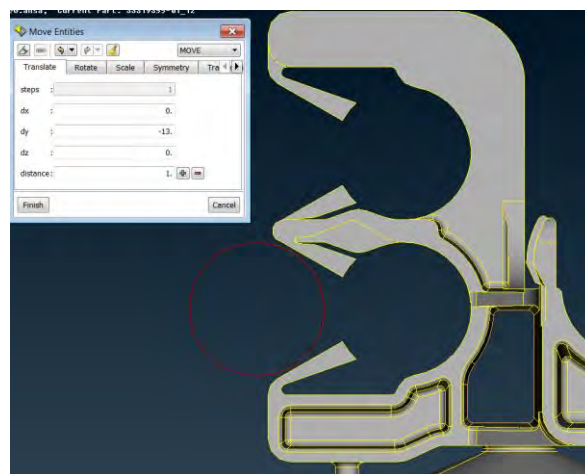


Figure 63. Move utility; Obtained from ANSA software.

9.6 Mesh

As explained in the Chapter 4, Section 4.3.2.5 Mesh, the user shall decide, based on his level of experience, the followed path to mesh the model. Manually or automatically.

Recommendation: It is highly recommended to use an automatic mesh as a base or reference one, for then fix manually the critical, difficult or specific parts of the model. Based on the experience acquired by the parameters study developed in Chapter 4, some mesh scenarios have been created in order to obtain an accurate and proper base meshing. Those are the following: *Clip-Shell.ansa*, *Clip-Solid.ansa*

Recommendation: If the user want to change mesh size, quality criteria or any other parameter, one can do it within the provided shell scenario.

Create automatic shell mesh

- Batch mesh tool → Read scenario → load files

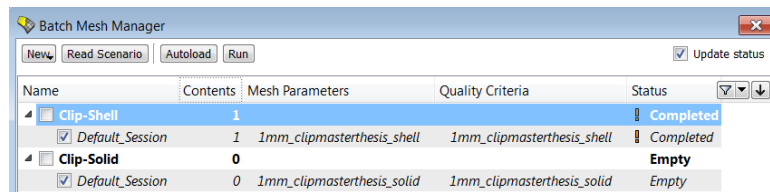


Figure 64. Batch mesh manager, Obtained from ANSA software.

Load the model parts to the mesh tool

- Contents → select pending items → Green right arrow

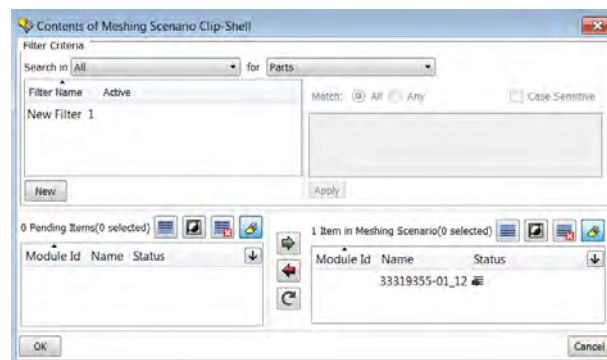


Figure 65. Contents meshing scenario; Obtained from ANSA software.

Run the shell meshing scenario

- Click the Shell scenario → Unclick the remaining → Run

Recommendation: The solid mesh is performed based on the shell mesh, so one may consider to improve and check the quality of the shell mesh in order to obtain a high-quality solid one.

In order to check the quality and perform an adequate mesh improvement, is necessary in the batch mesh window to:

- Right click Mesh parameters → Copy to global settings → Mesh and quality criteria

Recommendation: Coping the mesh and quality to global settings, the user is saving time in subsequent actions. When a new meshing tool will be used, the default parameters will be the copied ones from the scenario.

Recommendation: The hidden drawing style allows to go through all the elements out of the quality criteria range. See Figure 66.

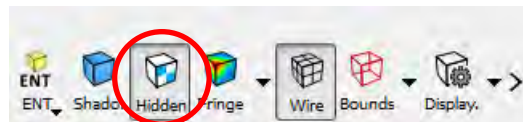


Figure 66. Hidden drawing style; Obtained from ANSA software.

Improve the mesh quality

- Hidden drawing style → Right click in off elements (see Figure 67) → Fix quality

Recommendation: The Fix quality option, allow the user reducing those issues in a fast and easy way. As explained before in the Chapter 4, Section 4.3.2.5 Mesh, the user should highly improve the shell mesh, until not obtain off elements based on its quality criteria.

Shell			
quads	:		0
trias	:		90596
total	:		90596
Off	:		135
Volume			
Unmeshed	:		1

Figure 67. Summary of elements in model; Obtained from ANSA software.

Recommendation: Once the shell is free of issues based on the quality criteria, a visual review is needed. Some parts in the model may need to be improved, as there are critical macro areas for the contacts or strain concentration, with not fine enough meshing. See Figure 68.

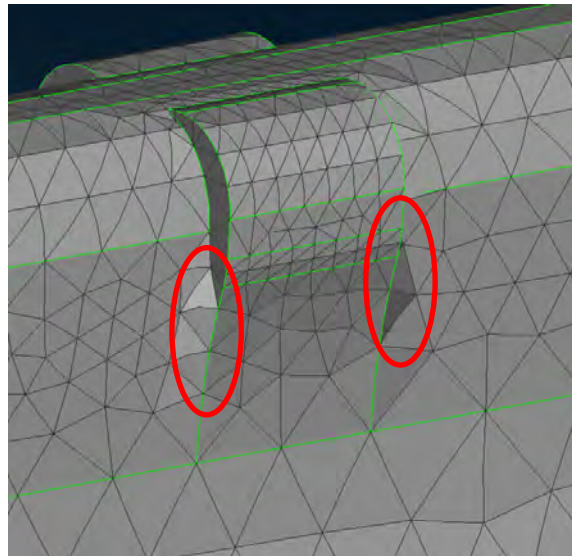


Figure 68. Critical mesh areas in the model; Obtained from ANSA software.

Recommendation: The user may use mainly two functions in order to improve the critical model areas, Reconstruct and Reshape. If the user deems it appropriate, one can use even more manual tools, like Num. +/-, Cut or Join.

The previous functions are briefly explained, in order to help the user to understand the benefits of each one:

- Reconstruct: Semi-automatic tool to produce high quality shell mesh. The perimeter nodes, nodes conforming the line, are not moved so the nodal number is not affected. It has a fix quality function embedded and is especially good for flanges, holes and fillets.
- Reshape: Very similar to the previous one, but with the additional ability of change the perimeters node, cut, join and align meshing areas.
- Num +/- : Purely manual function to change the number of nodes in perimeter segments.
- Cut: Purely manual function to cut meshed areas by nodes.
- Join: Purely manual function to join meshed areas, by deleting perimeter segments.

Recommendation: Reshape is the most advanced quality improvement function and the most recommended to use function, but sometimes is necessary to use in addition the purely manual functions.

Improve the mesh in certain critical areas

- Mesh module → Shell mesh → Reshape:
 - Element type = Ortho tria.
 - Target element length = between 0.1 and 0.5, depending on the feature
 - Minimum element length = see following recommendation

Maximum element length = see following recommendation

Recommendation: If the user needs a very fine mesh, one must greatly restrict these two parameters (Min. and Max. element length) around the target length value.

Create automatic solid mesh, running the solid meshing scenario

- Batch mesh tool → Click the solid scenario → Unclick the remaining → Run

The process after the solid meshing is the same as with the shell mesh. It is necessary a quality check and fixation, for which the most common function again is the “fix quality” function in the Volume mesh improve module.

Recommendation: If the user notice than some solid elements still not meet the quality criteria after using the “fix quality” function, an improvement of the shell mesh should be performed again. It means that the volume mesh will have to be done again, in the same way as previously.

9.7 Properties and materials

It is common that when the mesh is done, two different properties appear by default, one for the shell and other one for the solid. The way of assign a material to each part of the model is defining it in the MID (material ID) of the property, and link in that way both characteristics. See Chapter 4, Section 4.3.2.6 Properties and materials for more information.

Recommendation: The material shall be implemented after finishing the volume meshing. In order to obtain to achieved both properties, shell and solid, automatically.

Material

ANSA software defines a default material to the model than can be changed manually or update a file from the data library.

Load new material (the file used in this report is a confidential database from the CAE Volvo department)

- Material DB → List → Read DB → from file
- Select the desired material (PA66, MID=35010) → Copy to list → Update by IDs:
 - ✓ Overwrite names
 - ✓ Overwrite comments
 - ✓ Overwrite colours.

Recommendation: When the name of the desired material in the container changes, means that it is updated.

Recommended file:

C:/vcc/cae/backup/common2/spdrm/DM/LIBRARY_ITEMS/materials/abacus/91520_ansa_abacus_mat_polymers_and_foams.

Properties

Edit default shell properties

- Properties container → Shell:
 - MID = reference the ID of the material previously loaded = 35010
 - T = thickness = 0.01
 - Type = M3D_ = three-dimensional membrane

Recommendation: The visibility “light bulb” allows the user to check if the desired elements are taken properly. One can also right click on each property and perform the action “show only” as an alternative checking method.

Edit default solid properties

- Properties container → Auto detected volume:
 - MID = reference the ID of the material previously loaded = 35010
 - Type = C3D_ = three-dimensional solid

Recommendation: Edit the properties name in order to clarify the analysis procedure.

9.8 Sets

Recommendation: If the user has previously defined the group of nodes or elements as sets, will be much easier linking them with a BC or a Contact.

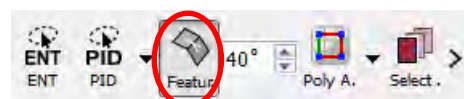
Each type of tool needs a different type of set, as explained below:

- Set container → New

Choose in the selection list the type of set (node, shell, solidfacet...) depending on the tool where it will be used:

- For BC: Node (Output as = set , output type = NSET)
- For contacts: Solidfacet (Output as = surface , output type = element)

Recommendation: In the Figure 69, the user see the tools for feature selection. It is highly recommended to use the remarked in red feature area tool. The user shall change the angle in order to tune the most suitable selection area.



*Figure 69. Feature selection tool;
Obtained from ANSA software.*

The following sets could have to be created in the simulation (some of them could appear or not, depending on the necessary contacts in each model):

- Fixed nodes Feet (see **Figure 18. Chapter 4, Section 4.3.2.7 BC**)
- Reference node studied tube (see Figure 70)
- Reference node aux. tube (see Figure 70)
- Contact hole 1
- Contact hole 2
- Contact Wing1 (Slave)
- Contact Wall 1 (Master)
- Contact Wing 2 (Slave)
- ...

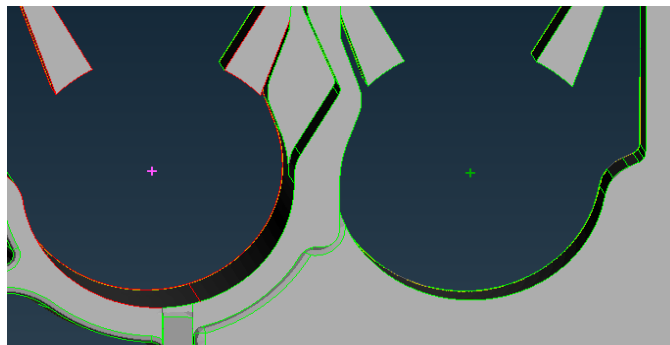


Figure 70. Tubes reference nodes; Obtained from ANSA software.

Recommendation: It is important to define the reference nodes for the tubes as Sets, so one is able to request outputs of them.

9.9 Boundary conditions

The user shall define two different type of BC, the fixation of the clip, and the necessary displacement of the tubes.

Feet

Recommendation: see the Chapter 4, Section 4.3.2.7 Boundary conditions, in order to understand the behaviour of the model with different BC.

Create the fixation of the clip in their feet

- Abaqus deck module → Boundary → Set → Fixed nodes feet:
 - Type = Fixed
 - format = direct
 - DOF = 123

Studied tube

Create insertion and pull-off displacement of the tube in the Y axis

- Abaqus deck module → Boundary → Set → Reference node tube:

- Type = Displacement
format = direct
DOF = 2
Magnitude = displacement explained in Chapter 4, Section 4.3.2.7,
Boundary conditions.

Restrict the studied tube in other directions

- Abaqus deck module → Boundary → Set → Reference node tube:
 - Type = Displacement
format = direct
DOF = 13456
Magnitude = 0

Auxiliary tube

Restrict the auxiliary tube in other directions

- Abaqus deck module → Boundary → Set → Reference node aux. tube:
 - Type = Displacement
format = direct
DOF = 1
Magnitude = 0

9.10 Contacts

The contacts are highly related with the friction and the surface interaction, see Chapter 4, Section 4.2.3.9 Contacts, that is why first will be defined those to important characteristics of the model: the friction and surface interactions.

Friction

Recommendation: As the reader previously read in Chapter 4, Section 4.3.2.11 Friction, the friction is a critical parameter which add complexity to the model. For this reason, only friction will be added in the critical areas for the study, avoiding friction in areas like inner wing area. The areas with no friction will be assigned to the default friction, which is zero.

Create friction for studied tube

- Abaqus deck module → Auxiliaries → Friction :
 - M = friction coefficient = 0.32 or 0.4 in this study

Surface interaction

Recommendation: The user shall create two surface interaction, one without friction and another one with the necessary friction for the study.

Create surface interaction without friction:

- Abaqus deck module → Auxiliaries → Contact → Surface interaction:
 - Friction = No
Method = Penalty linear
Clearance = 0
Scale factor = 0.05

Create surface interaction for friction:

- Abaqus deck module → Auxiliaries → Contact → Surface interaction:
 - Friction = Yes
 - Friction ID = identification number of the friction specified above
 - Method = Penalty linear
 - Clearance = 0
 - Scale factor = 0.05

Contact Tie

Recommendation: the user must be aware of the solidfacet size taken to perform the contact, the bigger the solidfacet the longer the running time due to the contact. As explained in the Chapter 4, Section 4.3.2.9 Contacts, the user shall use for this type of contact just the bottom part of the auxiliary tube.

Create contact tie in the auxiliary tube

- Abaqus deck module → Auxiliaries → Contact → List → New → TIE :
 - Slave = element
 - SSID = slave ID = previously defined in the Set module
 - Master = surface
 - MSID = master ID = previously defined in the Analytical rigid surface
 - Slave surface → Orient_S = Spos
 - Param = pos_toler
 - Distance = 0.3
 - Type tie coefficients = surface to surface

Contact Pair

Recommendation: This type must be implemented in the following interaction areas: wing/wall, studied tube/hole and aux. tube top part/wing. See Figure 26 Chapter 4, Section 4.3.2.9 Contacts.

Create contact pair in the studied tube

- Abaqus deck module → Auxiliaries → Contact → List → New → TIE :
 - Slave = element
 - SSID = slave ID = previously defined in the Set module
 - Master = surface
 - MSID = master ID = previously defined in the Analytical rigid surface
 - Interaction = previously defined in surface interaction, with friction
 - Slave surface → Orient_S = Spos
 - Param = pos_toler
 - Ext. zone = 0.1
 - Smooth = 0.2
 - Type tie coefficients = surface to surface

The remaining contacts pairs are defined in the same way, just changing the SSID and MSID by the respective set.

9.11 Step

Define the step

- Abaqus deck module → Auxiliaries → New
 - INC = 1000
 - NLGEOM = Yes = non-linear geometry
 - UNSYMM = Yes = non-symmetric solver
 - Analysis type = *Dynamic (see Chapter 2, Section 2.2.2 Linearity, nonlinearity and analysis type)

Recommendation: In order to help the reader to understand the behaviour in the analysis, following parameter definitions are explained:

INC= Maximum number of increments allowed. In case the analysis needs more steps, the analysis is not going to be completed.

TIME INC= Suggested time increment. The solver uses this time increment as the initial one. If the equilibrium force does not converged, the solver spends another iteration to solve it with a different time increment.

TIME PER= Total time of the step. The solver use this exact value. It is worth highlighting that this value, in concert with the value of the displacement, establish the specific velocity requested.

MNINC = Minimum time increment allowed.

MXINC = Maximum time increment allowed.

Define analysis parameters

- Step → Analysis parameters → Parameters:
 - Application = Quasi-static
 - TIME INC = 0.001
 - TIME PER = 0.1
 - MNINC = 1E-10
 - MXINC = empty

Inside the step, the user shall specify the requested outputs. As explained in the Chapter 4, Section 4.3.2.13 Output, the user must differentiate between history and field output. In FE there are different types of components (nodes, elements, etc.) and for each of those components there are specific outputs that can be requested. For example, the strains and the stresses can be only requested in the element, and the displacements, velocities and accelerations can be only requested on the nodes.

Recommendation: The user shall understand that with a bad set-up of the output requests, the post-processor will not be able to show them. The output parameters is where one can tell to the solver what is necessary to obtain.

Recommendation: Implement the following actions in the order explained below.

Define the output field requests:

- Step → OUTPUT requests:
 - Keyword = *OUTPUT
 - Parameter = Field
 - Frequency = 10
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *NODE OUTPUT
 - Identifying keys = RF, U, V
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *ELEMENT OUTPUT
 - Identifying keys = LE, S
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *CONTACT OUTPUT
 - Contact = both surfaces
 - Identifying keys = CFORCE, CSTATUS, CSTRESS
 - Click insert

Define the output history requests:

- Step → OUTPUT requests:
 - Keyword = *OUTPUT
 - Parameter = History
 - Frequency = 1
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *ENERGY OUTPUT
 - Identifying keys = ALLIE, ALLKE, ALLWK
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *NODE OUTPUT
 - NSET = reference node set of studied tube, explained in Section 9.8 Set.
 - Identifying keys = RF
 - Click insert
- Step → OUTPUT requests:
 - Keyword = *NODE OUTPUT
 - NSET = reference node set of auxiliary tube, explained in Section 9.8 Set.
 - Identifying keys = RF
 - Click insert

Identifying keys:

S	Stress components
PE	Plastic strain components
LE	Logarithmic strain components
U	Translation and rotations
V	Velocity
RF	Reaction forces and moments
CSTATUS	Contact status
CFORCE	Contact forces
CSTRESS	Contact stresses
CDISP	Contact displacement
ALLIE	All model internal energy
ALLKE	All model kinetic energy
ALLWK	All model external work

9.12 Export

Once the model is ready, the user need to create a suitable file for the software. This is the *.inp* file.

Export the model

- File → Output → Abaqus → Save *.inp* file
- Abaqus Output parameters → General:
 - Output format = 6.14
 - Output = Visible

Recommendation: In order to obtain a clean output file, the reader may click the option “Pre-output Model Check” with the “template = Abaqus checks”. This will check the entire model in order to filter any last minute issue.

9.13 Reading output files

Abaqus Solver gives several files to the feedback folder that provide different kind of information. The most important and useful files in this study are *.pre*, *.sta*, *.msg* and of course *.inp* and *.odb*, the input and output files respectively.

In simulation, it is common to obtain several errors during the analysis until one can get valid results. They can be divided into two types, the pre-processing error and the solving error.

- The first errors that could appear are the ones related with a bad pre-processing implementation, so the calculations has not even started. In case of having this type of errors, the Solver generates a faulty file with the format “*.odb_f*” a few minutes after sending the file to the Solver. The file *.pre* is also generated, and it shows the errors or warnings obtained with a small explanation and some tips

to solve it. The warnings can be ignored, while the errors need to be fixed by modifying those parameters in ANSA.

For instance the most typical errors of this type are distorted elements, due to a low meshing quality. If the user finds any Warning or Error due to distorted elements, it means that the mesh needs to be improved in those specific areas.

- In case of not having errors with the pre-processing settings, Abaqus generates the files *.msg* (complete analysis information) and *.sta* (a summary of all the job information) (see Chapter 4, Section 4.1.2 Computer solving). Both files contain several data such as the different increments, the iterations needed for solve each step, the cutbacks or all the time steps, but *.msg* file contains data in a more extended and complete way. The most typically issues found during a Quasi-Static (or dynamic implicit) analysis of second order elements are the huge time needed for the analysis (in comparison with first order elements) and the convergence.

At the bottom of those files (*.sta* and *.msg*) one can find the sentence “Analysis has been completed successfully” which means that the analysis is correctly done and there are no errors, even though, there could be some warnings. In other hand, “Analysis has not been completed successfully” means that the analysis has stopped due to convergence issues.

Recommendation: To solve them, one should change time parameters by examining at the *.sta* or *.msg* file. For example, it is common that the simulation stops in the beginning because there are so many cutbacks (represented with U). In this case, it is recommended to decrease the initial time increment (TIME INC) since there is a maximum of cutbacks allowed in each increment. When there is a cutback, Abaqus reduces the last time increment used in 25%, so if the initial increment time is too high it is not possible to reach the suitable value in 5 attempts. Another typical case is that the increment time starts to decrease so much, and there is need to give to Abaqus a higher minimum increment value (MNINC). Furthermore, one can find troubles when looking at the model in Abaqus Viewer, and having a look at *.sta* and *msg*. no errors or warnings appear and also there are no messages of “Analysis has been completed successfully” or “Analysis has not been completed successfully”, but the increments has suddenly stopped. If this happen, it is because of the queue that had been sent to was the wrong one for this model (short queue), so the suitable queue to send is the long one.

9.14 Visualization

Abaqus viewer is used in this project for the visualization of the results, but the reader could choose a different software if it is desired, as the visualization procedure would be practically the same.

Recommendation: Before starting the post-processing, the user may change some viewport annotation options, in order to obtain a better visualization of the results.

- Viewport → Viewport annotation options:

- Legend:
Set font → Size = 18
Numbers → Fixed
- Title block:
Set font → Size = 18
- State block:
Set font → Size = 18

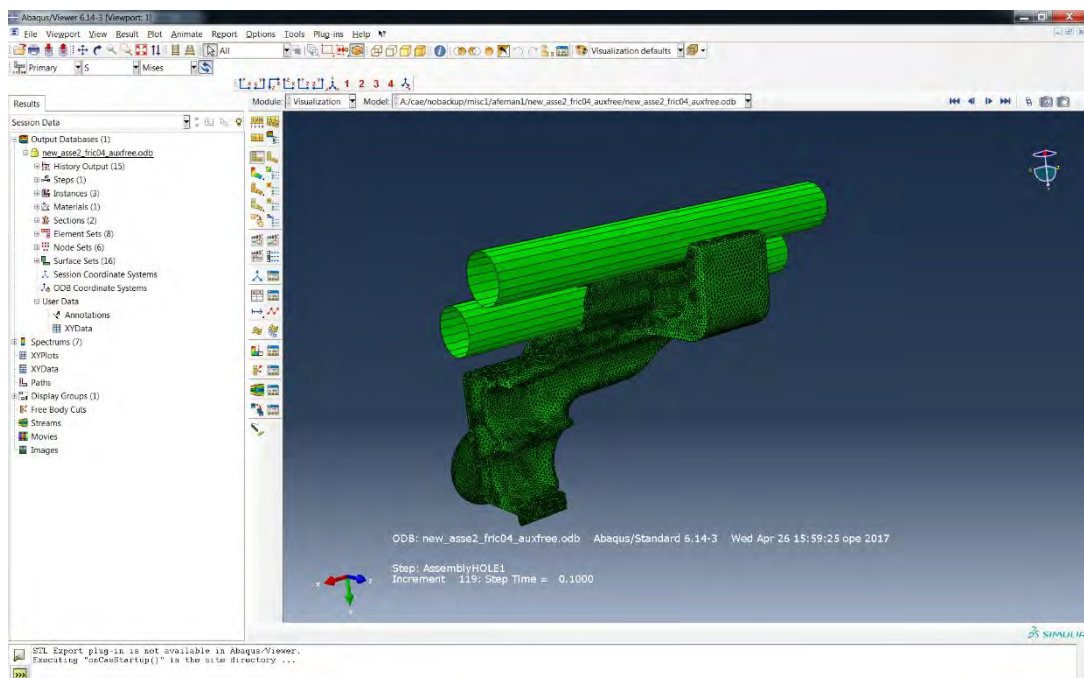


Figure 71. Abaqus viewer software overview; Obtained from Abaqus viewer software.

Open the solved file

File → open → *namefile.odb*

As previously the user had defined the history output for certain nodes set, now is possible to visualize their X-Y plot, this is the case for the reaction force (assembly and pull-off force) of the tube.

Recommendation: By obtaining the reaction force through the X-Y plot, the user is saving time and achieving more accurate results. The other option could be the visualization of the RF trough the contour plot, but it means wait for the loading of all

the model's frames and obtain a non-accurate result due to its output frequency of 10. See Chapter 4, Section 4.3.2.13 Output.

Visualize the X-Y plot of a variable

- Selection list of the model → History output → Variable (double click) (for instance: RF, Energy...)

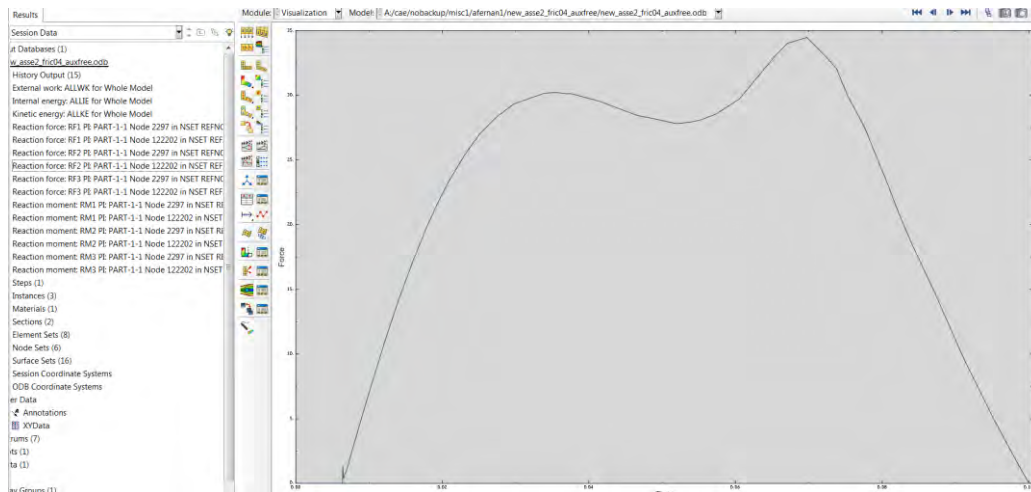


Figure 72. History output X-Y plot; Obtained from Abaqus viewer software.

Plot the deformed shape

- Plot → Deformed shape



Plot contours

- Plot → Contours → On deformed shape



- Choose the desired variable



Recommendation: The user must choose Max. principal logarithmic strain (LE) in the plot, in order to check the efforts in the model.

Recommendation: In Options → Contour the user can change some parameters as colours, style or limits of the plot.

Recommendation: When plotting the strain, a 75% average threshold is the default value. The default setting of 75% average threshold appears during the visualization of some parameters such as the strain. It refers to the discontinuity or continuity of the representation of the parameter, that is, if the software makes a softer representation by averaging the adjacent results. This percentage can be modified or deleted, and it means that the elements with less than 75% of strain similarity are averaged in order to get smoother and more continuous results. The elements with a similarity more than 75% are not averaged since the results are quite close.

Recommendation: by deleting this option, or what it is the same, or turning the percentage to 0%, one can obtain discontinuous results and consequently, more realistic results. On the other hand, one can apply a value of 100%, obtaining smoother results.



Figure 73. Explanation average threshold

Change averaging threshold value

Results → Options → Computation → Unclick Average element output at nodes.

The software allows the scale the deformations in order to magnify, reduce, or distort the deformed model shape (Abaqus/CAE User's Manual, 2012), this function is called "Scale factor".

Recommendation: The user should keep during the whole visualization process the scale factor in a value of one, in order to see the real behaviour of the model.

Set the Scale factor

- Options → Common → Basic → Deformation scale factor → Uniform = 1

The post-processing software is useful also for represent the obtained results in a clear and good looking way. For that is possible to create animations of model or prove some results values in the plotted model.

Recommendation: For create an animation the user must create it based on the time history.

Create animation:

- Animate → Time history → Save as:
 - Format = AVI
 - Capture = current viewport
 - ✓ Capture viewport backgrounds
 - Frame rate = 10

Recommendation: The user may take into account Light and Graphic options in order to obtain a proper animation. Those parameters can be checked in the Figure 74.

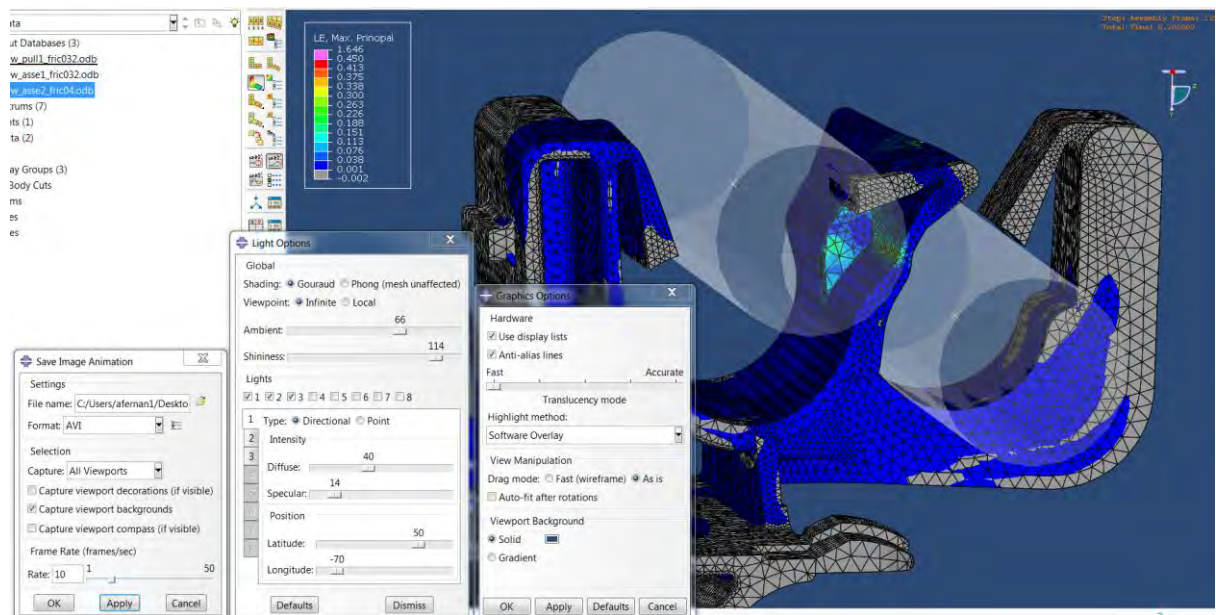


Figure 74. Animation options; Obtained from Abaqus viewer software.

Prove value: Function to check the value in specific point of the Plot or graph



10 Future work

There are so many next steps that could create improvements on the current simulation tool, but, until they are not implemented and compared there is no possible to foresee their good or bad behaviour. Some of those improvements could be:

- Change the fixation on the foot. The foot has been removed and it has been changed for a simplified model in this study. In order to make the model as real as possible, the foot should be kept, and it should be fixed by making a contact with the real surface that is on the car. Adding that extra foot-car contact will increase the complexity; the mesh needed for the foot should be really fine, the number of equations increase considerably, some convergence issues could appear due to the contacts, etc. Once this improvement is carried out there is need to check if it is worth it, since could happen that the results do not improve or the simulation turned really complicated.
- The friction coefficients has been selected in a theoretical way. To have a more realistic and accurate value it could be useful to make a friction test with the exact same materials and in the same conditions, and also establish different friction coefficients with the different conditions.
- Try other different type of contacts. The contact pair used at the beginning is the most realistic, but in complex models it does not work properly, so it has to be replaced by the tied contact. This contact works well and the behaviour is really realistic, but it could be good to try some other and see the difference.
- Other important improvement could be to change the mechanical properties of the polyamide in Abaqus, in order to imitate the changes on the properties of the real material with the different environmental conditions.
- Try the simulation method with other type of clips that have different shapes in order to check if the simulation tool described before could be effective in a general way, and also to improve it.

If the simulation is improved and it is checked that it works with other type of clips, the simulation tool could even replaces the real test as verification method for the technical requirement.

11 References

- Abaqus Analysis User's Guide. (2012)[ebook] Available at:
<https://www.3ds.com/products-services/simulia/support/documentation/>.
- Abaqus, Quasi-Static Analyses. (2005). 1st ed. [ebook] Available at:
<http://imechanica.org/files/15-quasi-static.pdf>.
- Abaqus/CAE User's Manual. (2012)[ebook] Available at:
<https://www.3ds.com/products-services/simulia/support/documentation/>.
- ANSA version 15.2.x User's Guide. (2015). [ebook] Available at: <https://www.beta-cae.com/documentation.htm>.
- Getting started with Abaqus: Interactive Edition. (2012). [ebook] Available at:
<https://www.3ds.com/products-services/simulia/support/documentation/>
- Hammar, M. (2012). *ABAQUS static and quasi-static analyses: Convergece and accuracy*.ppt. Available at: Volvo Car Corporation intranet.
- Imechanica.org. (2013). *Influence of Penalty stiffness factor on simulation results*. [Online] Available at: <http://imechanica.org/node/6768>.
- Jetpress.com. (2017). Cable Clips & Pipe Clips from JET PRESS. [Online] Available at: <http://www.jetpress.com/Products.aspx/12959/CableClipsPipeClips/>.
- Material data center. (2017). [online] Available at: Ref:
<http://www.materialdatacenter.com/ms/en/Scanamid/Polykemi+AB/SCANAMID+66+A33%5EE/ababbe12/1916>.
- Nafems.org. (2017). NAFEMS The Importance of Mesh Convergence - Part 1 engineering analysis and simulation - FEA, Finite Element Analysis, CFD, Computational Fluid Dynamics, and Simulation. [Online] Available at:
<https://www.nafems.org/join/resources/knowledgebase/001/>.
- Polykemi. (2017). *SCANAMID 66 (PA66) - Polykemi*. [Online] Available at:
<http://www.polykemi.se/scanamid-66-pa66/?lang=en>.
- Rodríguez, A. (2014). Análisis Técnico – Análisis de Elementos Finitos – FEA. [Online] AlbrodpulF1. Available at:
<https://albrodpulF1.wordpress.com/2014/03/09/analisis-analisis-de-elementos-finitos-fea/>.

Appendix

Appendix 1: Contribution report

Appendix 2: Full results real test

Appendix 1, Contribution report

The present study involves several and different responsibilities and tasks. In order to develop the investigation in a systematic, structured and effective way, each of the tasks has been assigned to one of the researches, according to their strengths and skills.

This, however, does not mean that each task has carried out independently, but each task is led by one researcher while receiving help, support and a continued exchange of information from the other group member, so that both researches are involved in the whole study.

The task division established is the following:

Mariana Fernández del Olmo Heres

- Post- processing of the models
 - Interpret and understand results
- Real test (Chapter 5.)
- Correlation of real results and simulation (Chapter 6.)
- Polyamide study
- Recommendation for future clip design (Chapter 7.)

Alejandro Fernández Schmidt

- Models of study (Chapter 3.)
- Pre-processing of the models (Chapter 4.)
- Technical recommendation for clips simulation (Chapter 9.)
- Conclusions (Chapter 8.)

Appendix 2, Full results real tests

Table 24. Pull-off test. RT and room RH. Old clip design.

Sample	Hole	F _{max} [N]
1	1	231.11
	2	148.07
	Feet	486.42
2	1	222.09
	2	149.51
	Feet	307.09
3	2	152.96
	1	130.79
	Feet	413.31
4	2	141.71
	1	177.40
	Feet	430.83
5	1	168.85
	2	111.97
	Feet	439.12
6	2	116.92
	1	91.79
	Feet	411.89

Table 25. Assembly test. RT and room RH. Old clip design.

Sample	Hole	F _{max} [N]
7	1	41.16
	2	47.86
8	1	48.64
	2	47.97
9	2	65.93
	1	67.30
10	2	47.21
	1	46.17
11	1	28.71
	2	46.12
12	2	44.67
	1	33.54

Table 26. Assembly test. RT and room RH. New clip design.

Sample	Hole	F _{max} [N]
1	1	47.91
	2	99.40
2	1	38.20
	2	80.80
3	2	52.87
	1	39.51
4	1	50.66
	2	72.00
5	2	75.34
	1	44.53
X	1	43.81
	2	67.26

Table 27. Pull-off test. RT and ORH. New clip design.

Sample	Hole	F _{max} [N]
1	3	192.79
	1	236.09
	2	159.28
	Feet	579.64
2	3	200.92
	2	206.27
	1	103.50
	Feet	638.02
3	4	332.04
	1	230.98
	2	178.96
	Feet	541.94
4	4	352.40
	2x	214.45
	1x	222.97
5	3	187.40
	2	183.57
	1	71.77
	Feet	518.36

Table 28. Pull-off test. -30°C and 0RH. New clip design.

Sample	Hole	F _{max} [N]
6	3	276.32
	1	297.11
	2	213.50
	Feet	619.54
7	3	26.33
	1	185.19
	2	265.00
	Feet	674.95
8	4	342.16
	2	239.01
	1	217.51
	Feet	549.15
9	4	389.52
	1	304.64
	2	210.58
	Feet	668.62
10	3	232.11
	2	233.39
	1	271.26
	Feet	748.61

Table 29. Pull-off test. +80°C and 0RH. New clip design.

Sample	Hole	F _{max} [N]
11	3	143.46
	1	117.16
	2	90.28
	Feet	362.60
12	3	93.11
	1	103.72
	2	94.57
	Feet	367.23
13	3	244.56
	1	122.17
	2	111.42
	Feet	411.79
14	4	243.68
	2	94.40
	1	68.26
	Feet	377.08

15	4	218.95
	2	95.51
	1	100.11
	Feet	368.40

Table 30. Pull-off test. RT and 95RH. New clip design.

Sample	Hole	F _{max} [N]
26	1	59.46
	2	53.77
	3	137.76
	Feet	374.49
27	2	50.78
	1	62.76
	4	199.07
	Feet	443.39
28	1	48.87
	2	48.72
	3	122.40
	Feet	348.25
29	3	95.09
	1	63.76
	2	45.17
	Feet	269.78
30	2	45.42
	1	54.47
	4	163.80
	Feet	228.01

Table 31. Pull-off force. -30°C and 95RH. New clip design.

Sample	Hole	F _{max} [N]
21	4	215.13
	2	99.96
	1	46.45
	Feet	697.64
22	3	141.46
	2x	173.30
	2	62.04
	1	106.84
	Feet	681.23

23	3	35.67
	1	63.42
	2	88.37
24	3	127.77
	1	105.44
	2	96.63
	Feet	623.88
25	4	316.77
	2	94.19
	1	81.40
	Feet	330.22

Table 32. Pull-off test. 80°C and 95RH. New clip design.

Sample	Hole	F_{max} [N]
16	4	126.50
	2	37.64
	1	39.27
	Feet	349.16
17	3	78.32
	1	44.07
	2	42.22
	Feet	217.82
18	4	147.80
	2	46.80
	1	46.22
	Feet	172.52
19	3	74.59
	1	43.74
	2	34.80
	Feet	235.29
20	4	139.44
	2	40.51
	1	43.21
	Feet	292.62