Design and simulations of test stand for collaboration with Hilti Diamond Coring System DD 150-U

In cooperation with HILTI and Technical University in Vienna

Author: Svetozara Boyadzhieva
Abstract

The aim of this master thesis is to design a test stand for a Hilti Diamond Coring System DD 150-U. The purpose of this stand is to be stable enough to carry out a variety of drilling processes and to allow Institute of Production Engineering and Laser Technology in Technical University in Vienna to conduct different tests and evaluations of the process. The first step is to create a detailed 3D model in SolidWorks 2013. It should eliminate the physical influence of workers and allow to investigate the drilling process in detail. This means that the drilling operation itself should be conducted without the intervention of human force. In other words the goal is to provide constant force of drilling and easier positioning of the tool according to the location of the hole to be drilled. The design should be suitable for drilling in concrete blocks with size 20x20x20 mm and should assure stable and repeatable operating conditions. As soon as the test stand is created the assembly will be tested with respect to its stiffness and stability. This will be done by creating FEM simulations of several critical parts and the whole test stand. The finite elements method (FEM) is a numerical analyses tool that is used for the calculation of the displacements, stresses and strains of those components under internal or external loads such as the force of drilling. These types of simulations are offering the possibility to lower development costs and to save time by eliminating the need of rework and prototypes. In order to provide a constant drilling force weights will be attached to the drilling machine.

Designen utformas så att den lämpar sig för borrning i betongblock med storleken 20x20x20 mm och den bör vara sådan att stabilitet och repeterbarhet särskilt under normala driftsförhållanden. Så snart provbänken är framtagen kommer den att testas vid montering för att utvärdera provbänkens egenskaper med avseende på styvhet och stabilitet. Detta kommer att ske genom att skapa FEM simuleringar av hela provbänken samt av flera kritiska delar. Finita element metoden (FEM) är ett numeriskt analysverktyg som används för beräkning av förskjutningar, spänningar och töjningar på komponenter vilka uppstår då intern eller extern belastning sker såsom vid borrning. Dessa typer av simuleringar erbjuder möjligheten att sänka utvecklingskostnaderna och spara tid genom att behovet av omarbetningar och prototyper elimineras. För att erhålla en konstant borrkraft kommer vikter att fästas i borrmaskinen.
I want to express my gratitude to the Institute for Production Engineering and Laser Technology at the Technical University of Vienna for their support and guidance throughout the duration of my master thesis project.

I would also like to thank Hilti who provided the financial funding for this project and giving me the opportunity to be a part of it.

Acknowledgement is also due to KTH Royal Institute of Technology in Stockholm who made my stay in Vienna during this period possible.
# Table of content

## Contents

- Abstract .......................................................................................................................... 1
- Sammanfattning ............................................................................................................... 2
- Acknowledgement .......................................................................................................... 3
- Table of content ............................................................................................................ 4
- Notation .......................................................................................................................... 6
- Standard Abbreviations ................................................................................................. 7
- List of figures ................................................................................................................ 8

### Chapter 1 ...................................................................................................................... 1
- 1.1 The company ........................................................................................................... 2
- 1.2 The University ......................................................................................................... 2
- 1.3 Need to improve the process of drilling ..................................................................... 2
- 1.4 The machine ............................................................................................................ 3
- 1.5 Test stand concept ................................................................................................... 4
- 1.6 Outline ...................................................................................................................... 6

### Chapter 2 ...................................................................................................................... 7
- 2.1 Objective ................................................................................................................ 8
- 2.2 Components ........................................................................................................... 9
- 2.2.1 Concrete block ................................................................................................... 9
- 2.2.2 Frame .................................................................................................................. 10
- 2.2.3 Euro pallet .......................................................................................................... 13
- 2.2.4 Support plate ...................................................................................................... 13
- 2.2.5 Inserts .................................................................................................................. 15
- 2.2.6 Frame walls ........................................................................................................ 16
- 2.2.7 Index Plate .......................................................................................................... 18
- 2.2.8 Drill stand ............................................................................................................ 19
- 2.2.9 Drilling Machine .................................................................................................. 21
- 2.2.10 Wheel mechanism ............................................................................................ 22
- 2.2.11 Mounting plate .................................................................................................. 25
Notation

Chapter 2

C  Circumference of the moving wheel
S  Distance between the wheel and the weight
D  Diameter
\( g \)  gravity force
F  the force applied during drilling
r  radius of the rotating wheel
\( r_{Gw} \)  radius of the gear wheel
\( x \)  Wight is kilograms
\( C_{Gw} \)  Circumference of the gear wheel
\( D_{GW} \)  Diameter of the gear wheel

Chapter 3

X, Y, Z  Axis
SX, SY, and SZ  Normal stresses
TXY, TXZ, …, TZY  Shear stresses
\( \sigma \)  Normal stress
\( \tau \)  Shear stress
L  Length of a component
Standard Abbreviations

DD 150-U  Hilti Diamond coring tool
DD-ST 150-U CLT  Drilling stand
DOF  Degrees of freedom
FEA  Finite element analysis
FEM  Finite element method
3D  3-dimentional
List of figures

1.1 Diamond Coring Tool DD 150-U ............................................................................................................. 3
1.2 Drilling Stand DD-ST 150-U CLT ........................................................................................................... 4
1.3 Test stand concept ................................................................................................................................. 5
2.1 Test stand design ...................................................................................................................................... 8
2.2 Concrete block with position of the holes ............................................................................................. 10
2.3 Preliminary sketch of the frame ........................................................................................................... 11
2.4 Sketch of the tube shape ....................................................................................................................... 11
2.5 3D model of the frame and the positions of the holes on it................................................................. 12
2.6 Euro Pallet and its dimensions ............................................................................................................ 13
2.7 Support plate and its location on in the frame ....................................................................................... 14
2.8 Different positions of the index plate and the drill stand. The holes on the support plate are coinciding with those of the stand and plate (shown with yellow) ................................................... 15
2.9 Inserts model and their position in one of the frame tubes............................................................... 15
2.10 Positioning of the walls with respect to the frame tubes .................................................................. 16
2.11 Bottom wall model and position with respect to the frame tubes .................................................... 17
2.12 Front wall model and position with respect to the frame tubes and the bottom wall .................. 17
2.13 Index plate model and its position in the test stand .......................................................................... 18
2.14 Model of the drill stand ....................................................................................................................... 19
2.15 Model of the Mounting device (Moving box) .................................................................................... 20
2.16 Model of the DD 150-U drilling machine ............................................................................................ 21
2.17 Tool model ......................................................................................................................................... 21
2.18 Basic idea of the wheel mechanism .................................................................................................... 22
2.19 Wheel model and its position on the moving box .............................................................................. 24
2.20 Connection of wheel and rope ........................................................................................................... 25
2.21 Mounting plate and its position .......................................................................................................... 25
2.22 Weight support walls and their position ............................................................................................ 26
2.23 Small frame and its position .............................................................................................................. 27
2.24 Close up of the position of the small frame ....................................................................................... 28
2.25 Linear guides and their position................................................................. 28
2.26 Positioning plate model and its location...................................................... 29
2.27 Holes location on the positioning plate ...................................................... 30
2.28 Support blocks and their position............................................................... 31
2.29 Threaded rods and their location ............................................................... 32
2.30 Overhead supports and their position ......................................................... 32
2.31 Encoder EC82-125-5 and its 3D model ....................................................... 33
2.32 Encoder system and its connection to the drill stand .................................... 33
2.33 Plastic connector for the encoder and its position ....................................... 34
2.34 Connecting cylinder ................................................................................. 35
3.1 Part with coarse and fine mesh .................................................................... 38
3.2 Typical stages in FEM simulations............................................................... 39
3.3 Degrees of freedom(DOF) ......................................................................... 39
3.4 Stress directions ........................................................................................... 40
3.5 Strain diagram ............................................................................................. 41
3.6 3D model for index plate simulation ........................................................... 42
3.7 Simulation tree for parts material ................................................................. 43
3.8 List of contact sets in index plate simulation .............................................. 43
3.9 Assignment of global contact set in index plate simulation ......................... 43
3.10 Geometry fixture if the index plate simulation .......................................... 44
3.11 On Cylindrical faces fixture ...................................................................... 45
3.12 Reference geometry fixture on the stand for index plate simulation .......... 46
3.13 Torque and gravity force on index plate simulation .................................... 47
3.14 Generation of mesh on index plate simulation ........................................... 47
3.15 Displacement in index plate simulation ...................................................... 48
3.16 Large displacement option ....................................................................... 48
3.17 Stress in index plate simulation .................................................................. 49
3.18 Material properties for plain carbon steel .................................................. 49
3.19 Displacement and strain results of index plate simulation ....................... 50
3.20 Factor of safety of index plate ..................................................................... 51
3.21 Highest position of the index plate ............................................................ 51
3.22 Stress results of second scenario ............................................................... 52
3.23 Displacement and strain results for the second scenario ........................................ 53
3.24 Factor of safety for the second scenario ............................................................. 53
3.25 Model for the frame simulation ........................................................................... 54
3.26 The model of support block for frame simulation .................................................. 55
3.27 Fixed geometry in frame simulation ..................................................................... 55
3.28 Cylindrical fixtures for the frame simulation ....................................................... 56
3.29 Reference geometry for frame simulation ............................................................. 56
3.30 Force acting over the mounting plate ................................................................... 57
3.31 Force representing the mass of the drilling machine and its stand ....................... 58
3.32 Torque direction in frame simulation .................................................................... 58
3.33 Mesh for frame simulation .................................................................................. 59
3.34 Stress results of frame simulation ........................................................................ 59
3.35 Concentrated stress areas in the frame simulation .............................................. 60
3.36 Displacement results from frame simulation ....................................................... 60
3.37 FOS in frame simulation ..................................................................................... 61
3.38 Second scenario model for frame simulation ......................................................... 62
3.39 Force and fixture application on the new scenario .............................................. 62
3.40 Stress and displacement results of second scenario ............................................. 63
3.41 FOS for second scenario ..................................................................................... 64
3.42 Test stand simulation model ................................................................................ 65
3.43 Fixtures and forces in the test stand simulation .................................................... 66
3.44 Contact sets for the screws and the overhead supports ....................................... 66
3.45 Simplified model of the linear guides .................................................................. 67
3.46 Mesh of the test stand simulation ......................................................................... 68
3.47 Close up of the mesh ......................................................................................... 68
3.48 Stress results of test stand simulation .................................................................. 69
3.49 Displacement results of test stand simulation ..................................................... 70
3.50 Strain results of test stand simulation .................................................................. 70
3.51 Model for the second test stand simulation scenario ............................................ 71
3.52 Forces applied in second scenario ...................................................................... 72
3.53 Stress results for second scenario ...................................................................... 72
3.54 Close up of the stress results in the bottom of the test stand .............................. 73
3.55 Displacement results for scenario two ................................................................. 73
3.56 Strain results for scenario two .............................................................................. 74
3.57 Second position where force is applied ................................................................. 74
3.58 Stress results for second hole location .................................................................. 75
3.59 Displacement results for second hole location .................................................... 75
3.60 Strain results for second hole location ................................................................. 76
3.61 Frame filled with sand .......................................................................................... 76
3.62 Test stand from the back ....................................................................................... 77
3.63 Test stand front view ............................................................................................ 77
3.64 Close up of linear guides ...................................................................................... 78
3.65 Mounted concrete block ....................................................................................... 78
Chapter 1
Introduction
1.1 The company

Hilti is a family owned company founded 1941 in Liechtenstein. Hilti develops, manufactures, and sells worldwide products for the construction industry. The company offers solutions for professionals in the field of measuring, in the field of drilling, demolition, cutting and grinding of concrete structures. Products are offered as well in the field of fastening and installation on concrete, steel and wood structures, and in insulation and protection. The company offers presales engineering services as well as repair, fleet management and life-time services. Hilti differentiate in its direct sales force from competitors. Hilti employs about 21,000 people in over 120 countries. Two thirds of its employees work in sales and engineering providing a direct contact to the customers. Hilti runs own production facilities in Central Europe, in the US, Mexico, India and China. The company has research and development centres in Europe and Asia. Drilling technology is part of the main product lineup of the company, providing new solutions for construction sites. Hilti mostly develops hand operated drilling machines and drilling tools for working with stone, concrete and bricks. The development in this field mainly focuses on improving the process, meaning to increase the drilling performance. [1]

1.2 The University

All the components of this master thesis were conducted at the laboratory for production engineering of the Institute for Production Engineering and Laser Technology (IFT) at the Technical University of Vienna. With the help and guidance of the university I was able to design, simulate and assemble a test stand.

The research at the Institute is engaged in the development of innovative manufacturing processes as well as of the necessary equipment technologies and production systems. For this reason IFT is considered to be one of the most important sites of production engineering research in Austria. [6]

1.3 Need to improve the process of drilling

The process of drilling has a great impact and importance in many sectors of the industry including automotive, construction, mechanical engineering, aerospace etc. It is one of the most elaborate, dangerous and costly operations. Despite being well known and developed process, there is a great need for more productive methods and improvements in the highly competitive globalized market. Manufactures are constantly working to produce holes with improved quality, to lower the working time and to increase the life of the tools. There are different factors influencing the outcome of drilling, depending on the type of drilling. The different types of drilling such as drilling in industrial production and drilling in buildings and civil engineering have different needs, base materials and technical requirements. The efficiency of drilling depends on the chips – in the case of metal or plastic machining; or on dust – in the case of
concrete materials. Excess of heat producing thermal deformations also has a huge impact on the outcome and it is often necessary to use large amounts of cooling fluids. Those factors combined with cutting force, cutting speed and feed-rate are the key variables responsible for good surface quality and extended tool life. It should be mentioned that drilling concrete is general done in two ways. The first one rotary hammer drilling is physically close to chiselling, and the second core drilling; the is close to grinding. In this work a core drilling system is used.

1.4 The machine

In order to explain why a certain machine was chosen it should be noted that it is not advisable to drill in to concrete with a standard drill. The difference between drilling materials like wood, metals and concrete is that the first two are fibrous or composed by layers. As for the concrete, it is a composite aggregate material, which means it requires one of two separate actions, hammering to break the aggregate or drilling to dig out the debris. Drilling in this work will be achieved using a Diamond coring tool DD 150-U shown in figure 1.1. It has 3 gears, the rotational speed gear 1 under no load is 780 turns/minute, the speed on gear 2 is 1520 tours/minute and on the third gear 2850 turns/minute can be reached.

The machine can be used for rig-based wet drilling in concrete for pipe penetrations, cable trays and ducts and for the installation of railings and barriers. It is also applicable as hand-guided wet drilling for setting large-diameter anchors and reinforcing bars, or hand-guided dry drilling in masonry for pipe penetrations in plumbing, heating and air conditioning installations and for chimney and stove installation.

![Figure 1.1 Hilti Diamond Coring Tool DD 150-U](image)

General features of the diamond coring tool:
- Rig-based or hand-held wet and dry drilling with one tool in different base materials without need to change rig components
- High motor performance and robust three-speed gearing for optimum drilling speed
- 360° adjustable side handle with integrated dust and water management functions
- Power control LED helps new users to achieve the optimum rate of drilling progress and maximum core bit life

In order to be able to drill holes with bigger diameter the machine will be used in combination with drilling stand DD-ST 150-U CLT. The stand is shown on figure 1.2 and it can be attached to a surface using either the four M10 leveling screws, the load carrying M20 screw, the vacuum mechanism or by combination of the different possibilities.

![Figure 1.2 Hilti Drilling Stand DD-ST 150-U CLT](image)

**1.5 Test stand concept**

In order to have a stable evaluation of different components of the drilling process many factors should be considered. The uncontrollable inputs and the deviations of the controlled ones should be reduced as much as possible. For this reason a basic test stand concept is created. This is the starting point of the detailed design which gives better visual representation of its purpose. Since drilling should be held without the influence of a worker all the components shall be assembled together. The most appropriate and not overly expensive solution is to create a simple frame to which all the components can be attached.

The next goal is to align the tip of the tool with the concrete block that shall be drilled and to provide possibility for changing the position of the drilling machine with respect to the number of holes that should be drilled. After careful consideration a basic system is created as shown on the figure below.
The test stand concept consists of few basic components:

- **Frame** – The frame consists of square tubes and will be filled with sand. This gives external stability of the whole assembly and allows us to connect the different components.

- **Work piece** – during experiments several same sized concrete blocks will be used.

- **Support** – It is used to fix the position of the work piece vertically and horizontally and to align it with the drilling machine.

- **Connecting plate** – It is used to connect the machine with the frame and also allows easier movement of the drill in vertical direction.

- **Drilling machine - DD 150-U**

- **Drilling stand - DD-ST 150-U CLT** - it is used for better accuracy and to prevent from undesired vibrations. Since a large amount of holes will be produced, the stand prevents from disturbances caused by physical fatigue of an operator. It will be also used for easier movement of the machine in horizontal direction.

- **Tool - diamond core drill of 20 mm diameter and 320 mm length**
The basic idea behind this test stand is to provide repeatable operating conditions and to allow easier and faster handling of the drilling process. Since severe number of holes will be drilled with it, it is essential for each component to be strong enough to withstand all the forces, weights and pressures during a sufficient amount of time. Each part is considered and designed in a way that drilling of a hole can be done without the interference of a worker and so that several components such as drilling force can remain constant. The frame is one of the basic and biggest components that hold together the whole assembly. Its size and shape will be determined according the needs mentioned previously. It should carry out the weight of all the components and assure stability and no unwanted vibrations caused by drilling. These considerations lead to the decision that the frame will be filled with sand. For this reason several walls should be designed and welded to it. The support will consist of several parts that allow easier changes or improvements of the general design according to future needs. The drilling machine and its stand are provided by Hilti, so they will be used as reference point for the design of the other parts.

1.6 Outline

Chapter 2: Test stand design – This chapter describes the design of the test stand. The model is created in SolidWorks 2013. Each component is analyzed in details and its purpose is explained. The chapter refers to the position of each part in the assembly and its connection with the other elements.

Chapter 3: FEM Simulations – In this chapter the test stand is being tested before being constructed. In the beginning there is a theoretical background which includes what is Finite Element Method and what are the steps to create such analysis in SolidWorks. By creating static simulations the displacements, stresses and strains of different components under internal or external loads are calculated. The overall stability of the assembly under the influence of drilling operation is also investigated. This kind of virtual testing allows optimizing and validating the designs in order to ensure good quality and safety, which is a very time and cost saving process.
Chapter 2
Test stand design
2.1 Objective

The design of the test stand is conducted in SolidWorks 2013. Each part is created according to the needed results. Designing a 3D model gives a lot of possibilities for experiments and testing of ideas. Several solutions were tried before creating the final test stand. The starting point for this project is the need to create a test stand that could be used for different types of data evaluations with defined drilling machine and drilling stand. The restrictions come from the size and characteristics of DD 150-U and DD-ST 150-U CLT. The design should be suitable for concrete blocks with size 20x20x20 with weight of approximately 18 kg each. The process is done using wet operation. The requirements and conditions for the system that should be created are:

- Application of constant force of around 200 N
- Preventing unwanted vibrations caused by the process of drilling
- Drilling of 16 holes on each block
- Easier and faster positioning of the tool according to the location of the hole to be drilled
- Possibilities to attach an encoder

The test stand should also give possibility for further studies and researches in order to estimate in which area improvements are possible. This means that it should be able to maintain stability even if components are changed and/or new equipment with different weight is used.

![Test stand design](image)

Figure 2.1 Test stand design

It should be noted that the frame box will be filled with sand for better stability and there will be a thin plastic cover beneath the concrete block that will protect all the surfaces from the water
used during drilling. The list below consists of all the parts in the test stand and brief explanation of their purpose:

- **Euro pallet** – Used for easier transportation of the assembly
- **Frame** – Holds together the assembly and provide steady and secure experiments
- **Insert in frame x2** – Provide thread for connecting screws and carry out weight
- **Side wall x2** – Welded to hold the sand inside
- **Back wall** – Welded to hold the sand inside
- **Front wall** – Welded to hold the sand inside
- **Bottom wall** – Welded to hold the sand inside
- **Support plate** – Carry out the weight of the drill stand
- **Index plate** – Allows vertical movement of the drilling machine
- **Drill stand** – Allows horizontal movement of the drilling machine
- **Drill machine**
- **Drilling tool**
- **Wheel** – Creates constant force by attaching weights to the system
- **Encoder assembly** – Used to attach an encoder to the test stand
- **Mounting plate** – Provides mounting area for different components over the frame
- **Weight supporting wall x2** – Carry the weight of the concrete block and ensures it is high enough with respect to the tool tip
- **Small frame** – Allows usage of linear guides
- **Linear guide x2** – Provide easier positioning of the concrete with respect to the tool tip
- **Positioning plate** – Provides surface for positioning of the concrete and attaching of cover
- **Support block x4** – Assure stable and exact location of the concrete
- **Threaded rod x4** – Used to attach the overhead supports
- **Overhead support x2** – Prevents the block from moving while being drilled
- **Concrete block**

### 2.2 Components

#### 2.2.1 Concrete block

For the purposes of this work several concrete blocks with dimensions 20x20x20 are used. The weight of one block is approximately 18 kg and each should be drilled 16 times. The figure below shows the basic model of the concrete block with the position of all the holes. The distance between the holes is 40mm horizontally and vertically. To be able to drill the holes on each block a system of movement is design. The location and placement of the drilling tool on the exact positions will be accomplished via the drill stand, the index plate and the support plate. The connection between the index plate and the frame is used for vertical alignment (see 1.2.7). The horizontal movement is realized by the attachment of the drill stand to different positions on
the index plate (see 1.2.8). As for the support plate, it is designed to assure stable grip of the stand in all the possible positions.

Since the concrete material can’t be found in SolidWorks library it is custom made. For each property of the concrete there is a known range of values. When choosing their magnitude for the purposes of the simulations only the lowest values are chosen. This decision is made because when applying force to the concrete the lowest values will lead to bigger deformation. On the other hand this deformation will lead up to higher influence of the drilling process over the whole assembly. The purpose of the FEM simulations that are conducted in chapter 3 is to evaluate the behavior of the components in the worst possible scenario. In this way the test stand can be designed in a better and more stable way. Since the concrete will be pushed by the drilling tool it is important to include the compressive strength in the properties and since it varies between 20 and 40 MPa, the value of 20 MPa is assigned to the model.

2.2.2 Frame

The first part that is designed is the frame. The reason for this decision is because the size and characteristics of the whole assembly depend on this component. It consists of tubes that are welded to each other. The first step of creating the model is to draw a 3D sketch, in which each line will represent one pipe. The parameters and distances between the tubes are defined on this stage, even before determining their size and shape. If the model needs some changes to fit with the other parts and/or needs a pipe to be added or removed, it can be done by going back in the sketch and conducting the alteration – the system will automatically upgrade the model. The height, width and length of the tubes were improved several times until it fits perfectly with the
other components. Also it gives the possibility for a worker to have easy access to all important and often used areas.

![Figure 2.3 Preliminary sketch of the frame](image)

The construction consists of several pipes with size 60x60 that are welded together creating a stable structure. The next step is to use the function `<Structural member>` that could be found in the menu `<Insert>` and then `<Weldments>`. A sketch, with dimensions as shown on the figure below, is used to create the shape of the tubes. This function is very useful in the sense of creating diagonal pipes. The system gives the option to choose the angle and the twist of each tube and the point where it is connected to the other members of the structure which removes the need to manually calculate its length.

![Figure 2.4 Sketch of the tube shape](image)
The ready model is shown on figure 2.4. To finish the part several holes should be placed on it. The first pair is on the front side, shown with red. They have diameter of 16 mm and go through both the walls of the tube. Their purpose is to connect the frame with the Index plate. Since the width of the walls is insufficient to hold a screw and to withstand the weight of the plate and the drilling machine, steel inserts are designed so that they can fit inside the tube. The distance of the right hole from the end of the pipe is 65 mm which is enough so that there is no collision of the Index plate with another part of the frame. Also, the space on the left is big enough so that the plate can be moved up and down (see 1.2.7). The holes shown with yellow are used for the connection bolts between the frame and the Mounting plate. Their exact position is determined by the size and characteristics of the plate on top (see 1.2.11). The diameter is again 16 mm but this time for a stable grip, a simple nut will be used underneath the pipe.

It should be noted that after the whole assembly is constructed and all the walls are welded, the frame will be filled with sand which will give even higher stability and will lower the influence of any kind of uncontrollable variables. The diagonal tubes that are created on the sides of the main square and the extra tubes on the bottom are used for better endurance of the walls that will be welded (see 1.2.6). The two long diagonal pipes brought out from the main square of the structure prevent from bending caused by the reaction force from drilling. They also provide resistance of the main square with respect to the weight of the index plate and everything attached to it.

After the model is finished the last step is to apply the material. For the frame a SolidWorks DIN Material is chosen, which is a Structural Steel 1.0038 (S235JRG2).
2.2.3 Euro pallet

A standard euro pallet is chosen for this set-up. Shown on the figure below is the general size of the pallet. This part isn’t designed in SolidWorks but it is still drawn for better vision and fully finished virtual assembly of the test stand. The euro pallet is wooden with size 1200×800×144 mm and it is nailed with special nails in prescribed pattern.

![Euro pallet diagram](image)

The pallet is generally used as a transport structure since it can carry a great amount of load. It is designed so that the load could be transported with a forklift or pallet jack. The dimensions of the pallet are chosen according to the size of the frame.

2.2.4 Support plate

The support plate is a simple plate designed to withstand the weight of the drill stand DD-ST 150-U CLT. It is known that the total mass of the drilling machine and its stand is 22 kg to which it will also be added external weights (see 1.2.10). So in order to secure the whole assembly and to assure a stiff grip that prevents from unwanted vibrations during drilling, the support plate is welded to the frame. The material that is chosen for this part is plain carbon steel which will provide the needed stiffness. On figure 2.7 is shown the position of the plate and its 3D model.
The length of the plate is 249.5 mm which is with 0.5 mm smaller than the distance between the two tubes it’s connected to. This decision is made because it is essential for the plate to be slightly smaller in order to fit in without any complications. Also this makes the process of welding easier for the worker.

It should be noted that the drill stand has a hole in the middle with height of 120 mm and width of 20mm through which it will be connected to the support plate via two-dimensional stud – M20 on one side and M12 on the other. For this reason as it can be seen on figure 2.10 the plate has two identical holes and each has two different diameters inside. The reason for this design is that the bigger diameter allows a nut to be placed inside, in a way that it can be moved in horizontal direction along the length of the hole. Its diameter is 19.2mm which allows the usage of a nut that has width across flats 19 mm. The smaller diameter of the hole is 13mm which allows the connecting stud to move freely while it is attached to the nut.

The distance between the two holes and the distance between the two surfaces shown in purple on the figure above, provide possibility of vertical movement of the drilling machine. As shown on figure 2.8, in all the possible positions of the stand and the index plate there is matching hole from the support plate. The yellow lines represent the outlines of the plate, so it can be seen that in the upper position of the index plate, the lower hole is used. As for the other 3 locations the higher hole of the support plate is used. And since the holes are long enough they cover all the four placements of the stand according to the index plate.
2.2.5 Inserts

As already mentioned the walls of the tubes used for the frame are only 4 mm thin, which means that there isn’t enough material for a thread that could hold the index plate. Also since the whole weight is concentrated over one tube there is a risk for heavy deformations. In order to handle those situations 2 inserts are inset in to the assembly. They have the same size and shape as the inner surface of the tube and the hole that goes through them has a thread with size M16. To prevent the dislocation of those components, they are welded to the tube. Figure 2.9 shows their exact position and the SolidWorks model. The material that is used for these parts is Structural Steel 1.0038(S235JRG2).
2.2.6 Frame walls

The walls are an essential part of the frame assembly because they strengthen the structure and provide a closed box that could be filled with sand. The material that is chosen for the walls is plain carbon steel, which could be found in SolidWorks list of materials. Each wall has a width of 1.5mm which is enough to withstand the pressure of the sand inside. The walls are welded to the frame with approximate bead size of 5mm. The two side walls and the back wall have a simple rectangle shape with size respectively 580x540 and 580x700. Their size is chosen to be such so that the end of each wall reaches exactly the middle of the tube that it is welded to as shown on figure 2.10. This provides enough support and sufficient area for the weld.

![Figure 2.50 Positioning of the walls with respect to the frame tubes](image)

The other two walls have a little bit more complicated design. The first one that will be presented is the bottom wall. The model and position of the wall can be seen on figure 2.11. Its general size follow the same rule about reaching the half of each tube which after simple calculation gives dimensions of 700x540 mm. What is different with this wall is that it has several cuts that allow it to fit on the bottom of the frame box without collision with any of the surrounding parts. Each cut is designed so that the wall has a 5mm distance from every vertical or diagonal pipe. This can be seen on the close up in the figure below. This space is used not only for easier assemble but also for the welding bead. The width of this wall is again 1.5 mm, but because the surface is very big, the weight of the sand will bend the part. For this reason two additional tubes that are symmetrically positioned under the wall are added.
And last but not least is the front wall. It is welded on the inside of the frame box and since the width of the support plate is less than the width of the tube, there is no actual contact between the plate and the wall. Its height is 515 mm which is the distance between the tubes on top and the bottom wall. Since there is no need for connection with the upper tubes around 1.5 mm distance is left between them and the wall for easier assemble. This part is welded to the two side tubes, which means that there is again distance left for a good fit and weld bead around the diagonal pipes. The difference here is that it is also welded to the wall beneath it.
2.2.7 Index Plate

The purpose of the index plate is to allow easy movement of the drill tool in vertical direction without the need of additional equipment or constant adjustment of the position of the tool tip in respect to the different concrete holes. It is also used as connection component between the drill stand and the frame. The positions of the connecting bolts of the stand require wide area, especially when considering that it has 4 different positions in horizontal direction. For this reason the index plate is designed to be broad enough to fulfill those needs but also to fit in the limited space between the frame tubes. Figure 2.13 represents the model of the Index plate and its position with respect to the frame.

![Index plate model and its position in the test stand](image)

The index plate has a simple and yet a very useful design. The general size of the plate is 540x550x10 mm and the appropriate material for it is plain carbon steel. Shown in red on the figure above is the mechanism that is used for movement in vertical direction. Also this is the only connection point between the frame and the plate. This means that the whole weight of the plate including the drilling machine and its stand will have to be moved together up and down. Moving more than 30 kg and in the same time screwing the bolts that hold it together is something that needs to be prevented. For this reason the holes are designed in a way that allows relocating of the assembly without fully removing the bolts. It is enough to loosen the screws so that the plate can freely be pushed in the needed direction, than the plate can be picked up until the position of the bolt is in the horizontal area of the hole. This is easily done because the bolt that is still attached to the frame will follow the path of the hole. When choosing the new position of the plate it is enough to slowly move it to the slot and release it. The design is constructed so that its weight will assure perfect positioning while the worker can easily tighten back the bolts. The size of the holes is slightly bigger to assure facilitated movement, which means that they have a diameter of 16.2 mm with respect to the used M16 bolts. The distance
between each of the four possible positions of the plate is 40 mm, which corresponds to the distance between the holes that should be drilled in the concrete block. As it can be seen on the figure above the initial position of the plate is on the far right end of the frame, leaving enough space to prevent from collision when the assembly is pushed to the left.

Shown with yellow on the figure are two pipes welded to the frame with diameter of 30 mm. They are used as grabbing device for the worker when moving the index plate up and down. It should be noted that since the vertical movement is done by moving the plate, the drill stand and the drilling machine all together, there will be no collision between the drilling tool and the upper part of the plate.

As mentioned before the horizontal movement will be realized by moving the drill stand with respect to the index plate. Shown in blue are the threaded holes used for this purpose. The distance between them is again 40 mm corresponding to the horizontal distance between the desirable positions of the concrete holes. The distance between the two sets of holes is 210 mm which is taken from the original drill stand. Moving to the right or to the left doesn’t require unassembling of the index plate.

Last but not least is the big hole in the middle that is shown with green on the figure. As mentioned previously the weight of the drilling machine will be concentrated on the big stud with dimensions – M20 and M12. From one side it is connected to the support plate and from the other to the drill stand. The design of the hole on the index plate is made so that all the possible positions on the bolt are located within it. This can be seen on figure 2.8.

### 2.2.8 Drill stand

The model of the drill stand is not an exact replica of the real part. The component consists of three parts that are assembled together – basis, lever and rack. The whole assembly is shown on the figure below.
The basis is represented by a simple rectangle with size $470 \times 240 \times 45$. The position and size of the holes in the upper part and in the middle correspond to the real stand. The slot where the lever is connected to the basis also has the same dimensions as the real model and it is a simple representation created for the purposes of the simulations.

The material that is chosen for the stand is aluminum but the mass density is changed. This is done because the model is very simple and basic so its weight doesn’t correspond to the original. In order to represent the model as close as possible to the real case, the components are designed to have the same weights. The mass of the components is as follows: basis 4.9 kg; lever 4.1 kg and mounting device (moving box) 4.6 kg.

The 3D model of the lever is very detailed as shown in the close up in the figure above. The measurements are taken from the real model and it has the same hollow shape. This is done because the model will be used later on for simulations.

The rack is taken from the library of SolidWorks where the exact parameters are introduced. It has module 2; face width 15 mm; pitch height 8 mm and length 800 mm. The rack is the only part from the stand that has a different material which is plain carbon steel.

![Figure 2.105 Model of the Mounting device (Moving box)](image)

Figure 2.15 represents the moving box which is a part of the drill stand ST 150-U CLT. The model is again simplified and it consists of a rectangle that has an extrude cut in the middle corresponding to the drilling machine model. What is interesting here is the presence of eight bulges that are designed to represent the wheels used for the movement of the box. Their purpose is to allow introduction of the mass distribution as close as possible to the real scenario. The weight of the drilling machine should be transferred to the lever only through those small contact areas and not along the whole surface of the movement device. Also there is a small rectangular cut that allows the rack to go through without collision. On the two sides there are bulges used for connection to the wheel mechanism and the encoder. Their position and size is exactly measured and represented in the 3D model. The last detail is that the curvature on top of the device corresponds exactly to the drilling machine model which allows the usage of simple mate option during assembly.
One of the main goals of the test stand is to assure drilling of holes through the whole length of the concrete blocks. Since the characteristics of the drill stand, the drilling machine and the tool give certain limitations, the position and design of many parts such as weight supports, mounting plate, positioning plate and index plate depend on them. Having 3D models that represent the general size and restrictions of those three components allows visualization and easier design of the other parts according to them.

### 2.2.9 Drilling Machine

The model of the drilling machine has similar curves and characteristics as the real component, but again it is not exact replica. The material used for this part is the same aluminum as previously but with different mass density so the model can have a mass of 8.2 kg. As mentioned before the curvature on the bottom corresponds to the one of the moving box so the connection between them can be conducted using mate options. What is important here is to create the hole used as handle because it will have big influence over the mass distribution during the dynamic simulation. Figure 2.16 shows the 3D model of DD 150-U Drilling machine.

![Figure 2.16 Model of the DD 150-U drilling machine](image)

The tool is designed as a different part and added to the drilling machine. After the real tool is measured the model is given a mass of 0.5 kg. It has outer diameter of 20 mm and inner diameter 17 mm. Figure 2.17 shows the tool model. The material used for this design is plain carbon steel.

![Figure 2.17 Tool model](image)
2.2.10 Wheel mechanism

One of the goals in this project is to provide constant force of drilling. For this reason the original mechanism used for movement of the drilling machine along the stand is removed and a new one is designed. The basic idea behind it is to have a wheel to which a metal rope with weights is attached. After positioning the drilling tool on the edge of the concrete, the weight will be pulled up manually and the rope will be rolled on the wheel. It should be designed so that the rope won’t unroll itself before drilling. After the machine is turned on the weight will lead to rotation of the gear wheel inside the moving box, by which the drilling process will start. The mass of the weights will provide constant force during each drill.

The size of the wheel and the mass of the weight are calculated according to the available space and the needed drilling force, which as shown on figure above is 200N. The distance between the center of the wheel and the lowest possible position of the load is represented by $S$. Let $D$ be the diameter and $C$ the circumference of the wheel. After measurement in the 3D model of the test stand, it is known that the distance between the pallet and the center of the wheel is $S=600\text{mm}$. Now from this distance the height of the tube on the bottom should be subtracted. This will prevent collision between the weights and the frame. It is also very important to measure the distance between the center of the wheel and the lowest point of the moving box. To escape unwanted contact between the drilling stand and the weights, the rope will not be fully rolled and this needs to be considered during calculations. Also because the weights will have certain height it should be subtracted from the total distance. Having in mind those parameters
the end safety distance will be taken to be 550mm. Using the drilling machine and the stand it is measured that 3.5 turns are needed in order to drill through the whole concrete block. Using those two variables the circumference can be calculated with the following formula:

$$C = \frac{S}{3,5} = \frac{550}{3,5} = 157,2\,mm;$$

The next step is to compute the diameter of the wheel using the well-known formula:

$$C = D \times \pi;$$

$$\Rightarrow D = \frac{C}{\pi} = \frac{157,2}{3,14};$$

$$\Rightarrow D = 50,06 \approx 50\,mm$$

$$r = 25\,mm$$

In order to find out how many kilograms the weight should be the following formula should be used:

$$F \times r_{GW} = x \times g \times r;$$

Where:
- g is the gravity force equal to 9.81
- F is the force applied during drilling
- r is the radius of the wheel
- r_{GW} is the radius of the gear wheel
- x is the weight in kilograms

It should be noted that the gear wheel that is inside the drilling stand allows movement along the rack. Since it is connected to the wheel, the torque in both of them will be the same. The circumference of the gear wheel is measured to be 65 mm, the calculation of its diameter is as follows:

$$D_{GW} = \frac{C_{GW}}{\pi} = \frac{65}{3,14};$$

$$\Rightarrow D_{GW} = 20,7\,mm$$

$$\Rightarrow r_{GW} = 10,35\,mm$$

Using these results the weight is calculated as follows:
After calculating the needed parameters a SolidWorks model is designed. As shown on the figure above the position of the wheel is on the left side of the moving box. The connection between them is very simple and its basic idea is to use a small metal cylinder with diameter corresponding to the small holes on both the components. When the parts are connected the holes should match as shown with red circle on the figure, so that the cylinder can pass through both of them. Its length should be equal to 20.5 mm so that it can fit in its position as shown on the sketch. A rubber will be inserted in the mortise with size 3.2 mm to hold the cylinder inside. This system will assure rotation of the gear wheel with the same torque applied from the weights.

As calculated the inner diameter of the rotating wheel is 50 mm and the size of the outer one is 70 mm, which prevents the rope from unrolling. The width of the wheel is also calculated so that when using a 3 mm steel rope there will be enough space to roll it 4 times without overlapping.

Figure 2.20 shows how the rope is attached to the wheel. There is a hole through which it goes to the upper surface. There it is attached with small steel rectangular that is screwed to the wheel. The screws can be loosened up which allows the rope to be removed and changed if needed.
2.2.11 Mounting plate

The purpose of the mounting plate can be recognized in its name. It is used for easier and better attachment of different components. In this scenario on top of the mounting plate there will be only weight supports, but it is designed so that there is enough surface that could be used for other type of components for further analysis or other types of investigations. The basic model consists only from several threaded holes that are situated in both ends of the plate. The material chosen for this part is plain carbon steel. The 3D model and its position in the assembly are shown on the figure below.
The size of the plate is 385x360x17. In this way it exceeds a little bit the size of the weight supporting walls on top which gives stability and resistivity. Since there are two components that the table should be attached to there are two pairs of holes. First there are four threaded holes with size M16 that are used for connection with the frame. They are shown with blue on the figure above and the head of the bolts will be placed on the bottom of the tubes. Since the total width of the frame and the plate is 77mm the bolts should not exceed this length to prevent from collision with the parts on top. Also it is important when filling in the frame with sand, to leave enough free space for screwing and unscrewing the bolts from beneath. On the other hand there are six more threaded holes with size M12 used for connection with the walls. The difference is that they have sockets for the head of the bolt on the bottom of the plate as shown on the close up in the figure above. Before connecting the plate with the frame it should be connected with the weight support walls because there won’t be any access to these holes later on.

2.2.12 Weight supporting walls

The purpose of the weight supporting walls is to bring the concrete block to the level of the drilling tool. After measuring the exact position of the tool and the height of the other components that are influencing the position of the block, it is calculated that the size of the walls should be 82.5x20x350. As seen on figure 2.22 the top of each wall has two threaded holes M10 which are deep 25 mm. They are used for connection with the small frame on top and their position is derived from this frame. On the bottom of the wall there are three M12 threaded holes which position determines the location of the walls with respect to the mounting plate. The distance between the two walls is carefully chosen because first it leaves enough space between them for other possible components and second it distributes the load of the concrete block over the tubes and not over the free floating areas of the mounting table. The middle of each wall coincides with the middle of the tube beneath it. The material chosen for the walls is plain carbon steel.

![Figure 2.13 Weight support walls and their position](image-url)
2.2.13 Small frame

The main purpose of the small frame is providing surface that could be used to inset linear guides. Its hollow shape is designed to match the support walls but also so that it won’t add unneeded weight. The figure below represents the model and its position in the assembly.

The small holes that are symmetrically placed on the four sides are used for connecting the linear guides. The distance between each hole is 60 mm which is taken from the original model of the guides. The size of the holes is M5 and they are threaded. On the inside of the frame there is an extrude along the edge with height 4 mm. It is used as restriction for the linear guides and the height is designed so there isn’t any collision. The four holes on the corners are with size M10 and are used for connection with the support walls. They have pockets for the heads of the bolts which provides flat surface on top, on which other components can be mounted. They are designed so that when locating the frame over the support walls their edges coincide as shown in figure 2.24. Since the mass distribution is very important the linear guides are located exactly over the support walls. For this reason in side direction the frame is wider. For safety reasons the edges of the plate are cut out which also gives a better finished look of the assembly. As it can be seen on figure 2.23 there are two more holes circled in blue, which have size M12 and their purpose is to hold the positioning plate symmetrical to the frame. This is the exact position of the plate and correspondingly of the concrete, needed for the force measuring tests. The material that is used for this part is plain carbon steel.
2.2.14 Linear guides

The linear guides that are used for this test stand are ordered from Mishmi. Respectively their 3D model is taken from their website [http://www.misumi-europe.com/](http://www.misumi-europe.com/) (18.06.2014)

The model that is chosen is Miniature Linear Guides Standard Blocks with assembly height 20mm, number of blocks 2, length 280 mm with WC set pair of rails. On each rail there are five M5 holes with pockets for the bolt heads. Each linear guide has two carriages with four holes that will be used to connect them with the positioning plate. The purpose of the linear guides is to allow different location of the positioning plate and respectively of the concrete block with respect to the drilling machine.
2.2.15 Positioning plate

The positioning plate is the final part that connects the concrete block with all the other components. Its main purpose is to locate the position of the block and the parts that are designed to hold it in one location.

As shown on figure 2.26 the plate consists of many holes distributed around the surface. On the bottom of the part there are two cuts. This is the area that will be in contact with the linear guide carriages and for this reason special surface finishing is applied. The process will lead to improvement of the surface texture and better appearance of the part. The depth of the cuts is 2 mm and the width is 50 mm on one side and 50.5 mm on the other. The reason for this difference of 0.5 mm is that it will provide easy and smooth movement of the linear guides. The general shape and size of the plate corresponds to the size of the small frame which leads to better safety conditions.

Figure 2.27 shows a close up of the upper face of the positioning plate with all the holes combined in groups by using different colors. The first group shown with red consists of sixteen holes with size M5 that are used for connection with the linear guides’ carriages. They have socket head cups which allow the positioning of different components on top without any collision with the bolts. The next group is colored in green and consists of four holes with size M16 that are used for the threaded rods that locate the concrete block (see 2.2.17). The distance between them is designed so that there is no collision between the screws and the concrete and in the same time to provide stability during drilling process.
The next group is colored in blue and there are four holes with size M12. They are used for mounting the force measuring device (see chapter 5). The position of the device should be as close as possible to the edge, so that the tool will have a good and easy reachable intersection with the measurement surface. The force measuring table is situated in the middle of the plate which also assures a good mass distribution and accurate evaluations. The yellow holes locate the position of the small blocks on top of the plate. As explained in 1.2.16 their purpose is to be holders of the block during drilling. There are four blocks with two connecting bolts each. The last pair of holes is circled with purple and consists of two M12’s with socket head cups. The purpose of those holes is to provide stable connection of the plate with the small frame. In this way the influence of the linear guides will be eliminated if needed. The small holes on the three edges of the plate that are not colored are designed for connection with a plastic cover. Since the drilling will be conducted with water assistance it is essential to cover the components and the sand. This also allows the usage of other equipment in the assembly, even if it includes electricity.

### 2.2.16 Support blocks

There are four support blocks designed to locate the concrete block and to prevent it from moving while being drilled. Since the concrete blocks will be changed several times it is essential to locate each of them at the same position. The design of the support blocks is very
simple; they are rectangular with size 15x15x50 and each has two threaded holes with size M5 and sockets for the bolt heads. The distance from the edges of the plate is chosen with respect to the position of the concrete block. It should be located 3mm from the front edge and exactly in the middle with respect to the side edges. In this way the concrete is as close as possible to the drilling machine but also has a very stable placement. The purpose of the support blocks is not only to locate the exact position but also to support the concrete while drilling force is applied. The material chosen for these components is plain carbon steel.

![Figure 2.2818 Support blocks and their position](image)

### 2.2.17 Threaded rods

In this assembly four rods with size M16 and length 250 mm will be used. They are used for the final and most stable support of the concrete. They are attached to the overhead support (see 1.2.18) and provide stationary and force resistant position of the concrete. Their height is calculated so that after being screwed in to the positioning plate they are still higher that the block allowing fixation of the supports on top. To assure a stable grip there is a nut on the bottom of each screw that tightens them to the plate. There are four more nuts used on the top to tighten the overhead supports to them. The position of each component is designed so that they do not collide with any other part and that they have sufficient distance from the other holes on the positioning plate. It is important to leave enough space between the screws in the back and the support blocks so that the nuts can be placed and reached whenever needed.
2.2.18 Overhead Supports

Figure 2.30 represents the model of the supports and their position in the test stand. The general idea behind those parts is to push down the concrete guaranteeing its stability and preventing from unwanted vibrations during drilling. The general size of one support is 280x30x10. For safety reasons and for better vision the edges have a fillet of 10mm. The holes have a diameter of 18 mm allowing easy and smooth penetration of the rods. On top of them as already mentioned are placed nuts that are tightened on the screws. The material that is chosen for these components is carbon steel.
2.2.19 Encoder system

An encoder is a device that converts linear or rotary displacement into digital or pulse signals. The most popular type of encoder is the optical encoder, which consists of a rotating disk, a light source, and a photo detector (light sensor). In this work, an incremental rotary encoder EC82-125-5 is used. It provides output when rotated and has the ability to change signals into information such as velocity or other motion related. After taking the appropriate measurements, a 3D model is created in SolidWorks as shown in the figure below.

![Encoder EC82-125-5 and its 3D model](image)

The model corresponds exactly to the original sensor and according to it an appropriate connecting system is designed. As mentioned before (see figure 2.19) on the two sides of the moving box on the drill stand, there are two mechanisms connected to the gear wheel inside. So when the weight is pulling down it creates a torque that rotates them simultaneously. This allows measurements to be taken by connecting the encoder to the right side of the stand. Using a plastic fixator and a steel connector, the rotation caused by the pulling force is translated to the encoder. This can be seen in figure 2.32.

![Encoder system and its connection to the drill stand](image)
The plastic cover is a hollow rectangular with size 40x60x65 and it is used to fix the main body of the encoder. On one side it has four small holes with size M3 which position corresponds to the distance between four of the holes on the encoder surface. In this way they can be screwed together which ensure that when rotating the wheel the main body of the sensor will remain stationary. On this surface there is one more hole with diameter of 10mm through which can go the rotating shaft. It is important that the hole has bigger diameter then the shaft so there is no contact area which will trouble the measurements. On the opposite side the hole has a diameter of 25 mm which allows it to fit with the stationary part of the box. As seen on the figure below the connection between them is realized by bolts that are going through both sides of the plastic cover fixing it to a stable position.

![Figure 2.22 Plastic connector for the encoder and its position](image)

Now we have a rotating shaft from the drill stand that should be connected to the shaft from the encoder. This is done by designing a cylinder that is attached to each shaft but doesn’t have contact with the plastic connector or the stationary surfaces of the encoder and the box. The diameter of the cylinder is 25 mm and as it can be seen on figure 2.34 on both sides it has holes with different sizes. On one side there is a hole with diameter 7 mm which corresponds to the shaft of the encoder and on the other side the diameter of the hole is 14 mm which fits with the rotating shaft of the box. The depth of the 14 mm hole is 13 mm which is enough for the shaft to go in but not to have collision between the connector and a stationary surface. Another specification of the design is the position of the two holes on the side of the connector. They both have size of M5 and are used for fixing the cylinder to the shafts. This is possible by screwing bolts which on the side of the box can go through the shaft providing rotation symmetrical to the gear wheel. On the other side the bolt will be tightened to the shaft surface of the encoder so that it is fixed and the rotation is transmitted. The material used for the connector is plain carbon steel.

During future evaluation of drilling parameter the encoder system will be covered with plastic bag ensuring that the water won’t damage the sensor.
2.2.20 Plastic cover

During experiments a plastic cover will be attached. It will prevent the water that is used as coolant during drilling, to damage any equipment, sensor or other parts. Its size will be enough to cover everything from the concrete block to the middle of the frame box. As mentioned in 1.2.15 the positioning plate has specific holes that will be used to attach the cover.
Chapter 3
FEM Simulations


3.1 Background

Finite Elements Method (FEM) is a numerical analyses tool used to calculate the displacements, stresses and strains of different components under internal or external loads. The main advantages are that it is a very time and cost saving process. For this reason the method has become a regular part of every design process, allowing engineers to escape the need of producing prototypes as well as delays and expenses caused by rework and damages. The analyses can be applied on arbitrary shapes and to any number of materials. There are numerous programmers that could be used for FEM simulations and in this work SolidWorks 2013 is used. FEM is also referred as finite element analyses (FEA) when it is applied for a specific fields such as stress, vibration or thermal analysis.

SolidWorks gives possibility for virtual testing and simulations allowing engineers to optimize and validate different designs in order to ensure good quality and safety. For the purposes of this work Static simulations are conducted and analyzed. The characteristics and endurance of different parts of the test stand and the whole assembly itself are investigated. This allows minimizing errors and providing stable solution. They are used for better understanding of the interaction between the different components, and also to test the overall stability of the test stand. Different parts and shapes are supported using the so called fixtures or restrains, while different forces pressures and torques are applied. The program represents the result with colorful displays, graphs and adjustable scales, which allows a quick and easy understanding of the problems. Displacements and stresses could also be animated and represented by automatic, user’s defined or true scale. SolidWorks offers variety of fixtures, boundaries and connections, on which can the applied sources such as forces, pressures, torque etc.

The basic idea behind FEM is to create a union of very big number of small shapes that represent a bigger complex shape. The smaller shapes are signified by triangles and are combined in a way that they model perfectly the original part. The process is called mesh generation and it is known that the smaller the mesh size is, the better and accurate results are indicated. This on the other hand has a negative aspect, because generating too many elements will lead to more calculations and the program will need too much time to solve the problem or it could generate errors in the values. However in complex assemblies different sized mesh can be applied to different component and areas of the parts, where bigger stress or displacement is expected. On the figure 3.1 is shown a rectangular part with coarse and fine mesh.
In order to carry through a full simulation several steps should be followed. As shown on figure 3.2 the first step is to determine the geometry and materials. Using a 3D model of a part or assembly is not enough to conduct a simulation. It is essential to insert material for each component, which will give information about elasticity, stiffness, weight etc. There is a possibility to choose from a library, which includes a big variety of materials (metals, woods, plastics etc.). However there is also an option where a new material can be included with manually filled in parameters. In some cases the program fails to read the characteristics from the original model and for this reason assigning the material in the tree of the study will prevent from errors. The contact sets are used to estimate the boundaries, contact surfaces and penetration options between different components. The fixtures, loads and pressures depend on the scenario that the engineer wants to investigate. In order for a mesh to be created the components should be from the same type. This means that when certain structural members are recognized as beams, the program is not able to mesh them simultaneously with the other components. A beam element is defined by a straight line connecting to joints at its end. It is usually created for frame structures that are welded together.

The last step is to run the simulation and analyze the results. It is recommendable to simplify the parts as much as possible to lower the process time. Each hole or complex shape leads to finer and more complicated mesh and if they aren’t an essential part of the experiment, they could be suppressed during the study. The program also offers the possibility to explore the factor of safety. When it is less than unity this indicates part failure. When the factor indicates low stresses in a region this means that there is a possibility to remove some material from there without harming the overall stability.
In the beginning of each simulation all the bodies are unconstrained and have six degrees of freedom (DOF) as shown on figure 3.3. Three of them are translational, which means movement along X, Y and Z axes. The other three are rotational again along the axes. When assigning fixtures between bodies some of the degrees of freedom disappear. SolidWorks also translates “Mates” that are made in the 3D model in the simulation study, creating constrains. This leaves them positioned in respect to each other regardless of any internal or external loads and pressures.
After completing a simulation several types of results can be analyzed:

**Stress results:** Stress is considered to be a measurement of internal forces in certain component which is caused by external loads. The unit of stress is force per unit area which means that the rough overall stress of an area can be calculated by dividing the force applied to it by the cross section. Stress is described by magnitude, direction, and the plane on which it acts. In simple words this means that if a force is acting in one of X, Y or Z axis, the stress values will be normal to that direction. So if a body is being pushed in X direction the stress value will be normal to it. However if forces are applied in X and Y direction this will result in stress normal to X and stress normal to Y. What is interesting here is that those forces will cause the material to try to slide and shear the body. The result of this is the presence of the so called shear stress. The figure below shows the different stress directions.

![Figure 3.24 Stress directions](image)

SX, SY, and SZ are the normal stresses and TXY, TXZ, ..., TZY are the shear stresses. Shear stresses are related by the following equations: TXY = TYX, TXZ = TZX, and TYZ = TZY.[3] The stress state at a point is completely defined by those six components:

**SX** - Stress in the X-direction acting normal to the YZ-plane  
**SY** - Stress in the Y-direction acting normal to the XZ-plane  
**SZ** - Stress in the Z-direction acting normal to the XY-plane  
**TXY** - Stress in the Y-direction acting on the plane normal to the X-direction (YZ-plane)  
**TYX** - Stress in the X-direction acting on the plane normal to the Y-direction (XZ-plane)  
**TXZ** - Stress in the Z-direction acting on the plane normal to the X-direction (YZ-plane)  
**TZX** - Stress in the X-direction acting on the plane normal to the Z-direction (XY-plane)  
**TYZ** - Stress in the Z-direction acting on the plane normal to the Y-direction (XZ-plane)  
**TZY** - Stress in the Y-direction acting on the plane normal to the Z-direction (XY-plane)[3]

In this work the equivalent Stress (also called von Mises Stress) is used to evaluate different components. It provides information to assess the safety of the design for many ductile materials. The difference in the von Mises stress is that it has no direction and it is defined by magnitude with stress units. To calculate the factors of safety at different points, the program
uses the von Mises Yield Criterion, which states that a material starts to yield at a point when the equivalent stress reaches the yield strength of the material. In simple words the strength of a material is given by yield strength. For example since the biggest part of the parts designed in this work are made of plain carbon steel, the material property is with yield strength of 220 N/mm^2. This means that at that exact stress value the material will begin to yield.

The equation that stands behind the equivalent stress was created by Von Mises and is as follows:

\[
\sigma = \sqrt{0.5 \left[ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 \right]} + \sqrt{3 \left( \tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2 \right)}
\]

where \(\sigma\) represents the normal stresses and \(\tau\) corresponds to the shear stresses. The basic idea behind this equation is that it combines all the stress values and plots a single “Von Mises Stress Value”. If it is higher than the yield strength the component is failing and if it is lower the part is safe and strong enough to withstand the loads and forces.[5]

**Strain results:** Strain is the ratio of change in length to the original length, or \(\delta L / L\). Strain is a dimensionless quantity. [5]

![Strain Diagram](image)

**Displacement results:** This represents the displacement of the components relative to the original geometry. It allows visual representation of the changed geometry and results will be given in mm.

**Factor of safety (FOS):** It is important to know that a factor of safety less than 1.0 indicates that the material has failed; factor of safety of 1.0 indicates that the material has just started to fail and a factor of safety larger than 1.0 indicates that the material is safe. These results are used to identify weak areas of a design or to find places where material can be removed without influencing the safety of the component.
3.2 Simulation of different parts

3.2.1 Index plate

In order to simulate the reliability of the index plate a separate simulation is conducted. In this case a full 3D model of the plate is used where all the holes are present. A new assembly is created containing all the parts that have direct contact with the plate. This includes three tubes from the frame, the support block welded to the frame and the drill stand. For the purposes of this investigation a simple model of Hilti stand DD150-U is created. It consists of the basic block with the same dimensions as the stand itself on which the force of drilling will be applied. The holes for the connecting bolts are also present in order to have more accurate connections between the components. It should be considered that there are 16 scenarios. They include four different positions of the plate with respect to the frame, and four possibilities to move the stand according to the plate. In order to better understand the stiffness of the plate several scenarios will be simulated. The basic 3D model of the assembly that is used for those simulations is shown on figure 3.6.

After starting a new static study the first step is to specify the material of each part. The index plate is made of plain carbon steel and all the other components are assigned to be rigid. Bodies are treated as rigid when they are much stiffer than its surrounding or when there is a big assembly and the solid is very far from the region of interest. In our case the frame, the stand and the support block are treated as rigid, because they are used only for support and force application and are not part of the needed results. The main reason and advantage is that the
simulation is significantly faster, since it save time by only making calculations for the deformable body. On figure 3.7 is shown the extended part tree of the simulation. The materials are applied from SoldWorks library and the green tick on the top of the tree shows that the material specification is complete. In this part of the project the stand is assigned to be made of aluminium, but its mass density is changed again in order to have the same weight as the original model.

![Figure 3.7 Simulation tree for parts material](image)

The next step is to apply a global Contact set which prevents penetration between the different components. By using the whole assembly in the component box, the program can automatically find the contact surfaces. It is also possible to manually select faces and assign sets between them.

![Figure 3.8 Assignment of global contact set for index plate simulation](image)

![Figure 3.9 List of contact sets for index plate simulation](image)

Now it is time to constrain the geometry by applying different types of fixtures. The first fixture is a standard fixed geometry and it is used to limit the movement of the frame and the support. In this way the whole assembly is stationary according to the original test stand model.
Since the tubes are separate parts, a face of each of them should be selected. On figure 3.10 are shown the characteristics of the fixture and the selected faces which in the model are colored in yellow. The close up in the bottom right angle shows the direction in which movement is prevented, in this case it is both x, y and z directions.

To escape the need to create solid bolts and enlarge the assembly and the needed calculations, virtual bolts are created using advanced fixtures. The method of putting restraints on the bolt holes is used since it simplifies the model and saves efficient time, but still leads to the same result accuracy. The first step is to create on cylindrical face fixtures that can be found under the advanced fixture options. It is important to select all the cylindrical. For example the top tube from the plane has two walls through which the bolt will go, which make two separate faces that should be selected. Figure 3.11 shows in yellow the on cylindrical faces, which after selection are restricted in radial direction. Also there is a close up of the arrows showing the direction of the restriction. There is an option to specify the amount of translations and by leaving it to zero the face is fixed to that exact location. In other words what has been done with this function is to tell the system that those holes can’t move anywhere in the radial direction with respect to each of their own axes. This replicates the presence of a bolt shaft.
For the next step split faces are created on the index plate and the drill stand. This is done by creating circular sketches around each hole on those components and using the split line function. In this way every individual area can be selected without changing the geometry of the parts. Those areas will be used to simulate the bolts heads. This is done on two steps because there are two separate parts with different reference faces. The first part is the drill stand, where the selected faces are shown in yellow - figure 3.12. The function that is used is reference geometry. The three split faces have restricted movement according to the top plane of the stand, shown in pink. From the translation options the normal direction is chosen. Having a value of zero doesn’t allow movement back and forth with respect to the index plate. The close up in the figure shows the direction of the fixture. Since the frame and the support plate are fully fixed and presented as rigid bodies, there is no need to create similar functions for the nut position. The second step is to use the same method to restrain the plate according to the frame. The shape of the split area can also be seen on the figure below.
After the input of all the needed fixtures is done, it is time to create the loads. Gravity of 9.81 m/s$^2$ is applied by choosing the appropriate plane for direction. The red arrow on figure 3.13 represents the gravity. Then by using the external loads menu a torque is applied. It is known that every action has an equal and opposite reaction. For this reason when drilling with constant force of 200N a reaction force with the same size will act on the drill stand. Since the stand is connected to the index plane with one bolt in the middle, the acting force will be in the form of torque. This means that the plate will push itself in the frame on the bottom and pull itself away from the top, where it is connected with two more bolts. In order to simulate this force an axes is created. It goes through the center of the hole for the bolt as shown on the figure below. By selecting the top face of the stand and the axes a torque is created. The size is 200N and the reverse direction option is used to specify the direction of pull and push. It can be seen on the figure that the arrows representing the torque change their direction with respect to the axes. Since the stand has two bolt holes on the top, the force is concentrated around them representing the tension created over the bolts.
The final step before running the simulation is to create the mesh. The scale used for the mesh is automatically generated by the program since there is no need for finer density. The result is shown on figure number. It can be seen that around every hole the program generates more complex and finer mesh. This leads to better accuracy, but also need longer time to calculate.
After running the simulation, the program generates results for stress, displacement and strain. Shown below on figure 3.15 is the displacement of the index plate in true scale and in automatically chosen scale. True scale means that the actual displacement can be seen if present. As shown on the left the plate stiffness is enough to prevent from actual deformation. The scale shows the amount of displacement in mm.

![Figure 3.15 Displacement in index plate simulation](image)

By analyzing the affected areas on the part itself, it can be concluded that the biggest influence over the deformation is the gravity. The index plate is sagging under its own weight. When its own weight is applied suddenly the steel bends very easily. The solution is to apply gravity and loads slowly in steps and update the stiffness each step. In this way the stiffness properly accounts the tensile stresses, so they will provide some resistance to bending. The small displacement solver, which is used by default, applies all loads including gravity to full value. The way to fix this is by using the option large displacement, which can be found by right clicking on the study name and choosing properties.

![Figure 3.16 Large displacement option](image)
The program increments the loads in steps, which means that in the first step 10% of the loads are applied after which deformations are solved. Then the stiffness matrix is updated and the loads are increased to 20%. The system continues updating the stiffness matrix after each increase by 10%. This method leads to longer analyses but the results are more accurate. It can be observed now the true influence of the drilling force over the index plate. On figure 3.17 is shown the stress results with true scale in front and back view. True scale means that the deformations scale has a value of 1. The von Mises scale is in N/m² because it is so little that when changed to MPa the values range from zero to zero. The minimum and maximum values are also plotted on the figures.

![Stress in index plate simulation](image)

**Figure 3.17 Stress in index plate simulation**

The stress is high in the area between the stand and the frame, where the force is applied. The maximum value is 0.431N/m² but it is also known that the yield strength of the steel used for the index plate is 220.59N/mm². Knowing these to values it can be concluded that the stress is not sufficient enough to need a change in the construction of the index plate.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Modulus</td>
<td>210000</td>
<td>N/mm²²</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.28</td>
<td>N/A</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>79000</td>
<td>N/mm²²</td>
</tr>
<tr>
<td>Density</td>
<td>7800</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>399.83</td>
<td>N/mm²²</td>
</tr>
<tr>
<td>Compressive Strength in X</td>
<td>N/mm²²</td>
<td></td>
</tr>
<tr>
<td>Yield Strength</td>
<td>220.59</td>
<td>N/mm²²</td>
</tr>
<tr>
<td>Thermal Expansion Coefficient</td>
<td>1.3e-005</td>
<td>K</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>43</td>
<td>W/(m·K)</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>440</td>
<td>J/(kg·K)</td>
</tr>
<tr>
<td>Material Damping Ratio</td>
<td>N/A</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 3.18 Material properties for plain carbon steel**
The displacement and the strain are shown on figure 3.19. Both of them are represented in true scale. Since the results are so low, there is assurance that the geometry will stay unchanged during experiment. The strain represents the ratio of change in length during and after load was applied.

In the result options there is a possibility to choose one more method for investigation. The factor of safety evaluates the values of safety at each node. This method is used to study the possibilities of part failure. When the value is bigger the area is considered safe. In this case the factor of safety is bigger than 20 which indicate that material can be saved by being removed. For the purposes of the evaluations the will be made with this part and the possibilities of adding and/or changing components, the index plate will remain with the same characteristics. Also this will prevent of damages that may occur is the size of the drilling force is enlarged. The result is shown on figure number 3.20.
As mentioned before several scenarios are investigated. Since the movement of the plate up and down applies big changes to the whole assembly, a new simulation is made. As shown in figure 3.21 the plate on the highest position has new critical areas. The face that is in contact with the fixture is relatively smaller and the position of the bolt on the stand is changed. This will also lead to different distribution of the torque. In the previous case the position of the stand itself was in one of the end positions. In this case the stand will be in one of the middle positions because the force that is applied from the stand will have less intersection with the frame. This may lead to bigger displacement and higher stress and for this reason the scenario is simulated separately.
In order to create this simulation the same restrictions are applied. The fixtures and the loads are duplicated. One of the changes that needs to be done in the simulation itself is to change the position of the axis going through the connecting bolt between the stand and the plate. This will lead to change in the distribution of the force. As it can be seen on the figure above the new split faces are conducted, according the new positions of the bolts. The old ones are suppressed in order to have simplified and easier to work with model. The mesh that is used corresponds to the one in the previous case.

Figure 3.22 shows the stress results in N/m². The first thing that can be noticed is that the results are in even smaller range. The values range until 0.072 N/m² and in the previous case the upper limit was 0.558 N/m². This is because the area where the force pushes the stand against the plate is smaller. Because of that the deformations are spread around the plate and for this reason the concentrated stress is decreased. Because of the changed position of the drill stand the stress values are now concentrated more around the support block that is welded to the frame. Figure 3.23 represents the displacement and the strain of this scenario in true scale. It can be noticed that here the range is also different. The displacement maximum value is 1.459e-011 and in the previous case it was 1.643e-010. The areas affected by displacement are bigger and the values
are relatively smaller. The values of the strain are also smaller but have approximately the same concentration. Still the values are insufficient to cause permeant or fatal damages.

The conclusion is that despite the change of the position of different components the end results are approximately the same. The distribution of the stress and displacement is different but the overall stability of the index plate remains the same. This can be easily observed by introducing the safety factor. As shown on figure number the safety factor exceeds 20. This means that even when the position of different components is changed, the index plate is still considered as safe.
3.2.2 Frame

One of the most important components that should be tested is the reliability and stiffness of the frame. This is the base of the whole test stand and it should withstand all the forces and the weight of each component. For this reason special attention is paid on the simulation of the test stand assembly. The basic 3D model used for this simulation is shown on figure 3.25 and it consists of all the tubes from the frame, a simplified model of the index plate, the support block welded to the frame, the inserts in the upper tube of the frame and the mounting plate. In the original model of the test stand the frame is made as beam construction. This means that it is created from one drawing and is in the form of one part. In this simulation a new assembly is constructed using separate parts for each tube. This is done because in a static simulation the mesh and the results of beams cannot be created simultaneously with other parts. Since we want to apply torque and force on exact surface areas there is a need to include the index plate and the mounting table in this scenario. A new 3D model of the index plate is created for the purposes of this simulation. Only three holes are created for the connecting bolts. Everything else that is not included is not essential for the results and will only overwhelm the calculations. Using split line, circular surfaces used for bolt heads imitation are created, as well as an area with the same size and position as the drill stand. This area is used for the torque applied by the stand, but since the index plate will be considered as a rigid body, the presence of the stand itself is not necessary. The purpose of the mounting table on top is to check the reliability of the frame when having a lot of heavy equipment on top. The weight of the structure that should be mounted is added with some additional weight that represents the possible change or improvements of components for further investigations and it is transferred to Newton. This gives a total value of 600N. All the holes are removed since they are not relevant and it will save time for the simulation.
When starting the simulation, each tubes material is defined as structural steel 1.0038 (S235JRG2). Then they are created separately in order correspond exactly to the original model. The exact length of the diagonal pipes and their surface area of the contact edges are measured from the beam structure created in the first model. Normal mates are used in order to represent the welding between them. The support block that is welded to the frame is also present and is defined as plain carbon steel. The 3D model is also constructed for the purposes of this investigation. It has the same size and parameters but the characteristics of the holes are changed. In this case there is only one hole, having the size of the bolt used to attach the stand to it. Using split line a circle around it is created to imitate the bolt head. The reason for this decision is that in this scenario the block won’t be assigned with fixed geometry. Because the result for the deformation of the frame is the main goal, it is important to represent every connection involved with it as close as possible to the real model. The force applied by the Index plate depends on the type of connection that it has with the frame itself. Picture of the block can be seen below.

![Figure 3.26 The model of support block for frame simulation](image)

After applying all materials the index plate and the mounting plate are assigned to be rigid bodies because they are not part of the desired results. As in the previous simulations a global contact set is applied, preventing penetration of different components in to each other. By choosing the right plane and direction a gravity force is added to the scenario. The first constrain that is applied is fixed geometry on all the bottom tubes of the frame. This won’t allow the construction to move in any of the direction. In the real model it will be achieved by filling in the frame with sand.

![Figure 3.27 Fixed geometry in frame simulation](image)
All the other fixtures consist of on cylindrical faces and reference geometry. It should be noted that the inserts are also present in the simulation and their original 3D model is used. The connection between the index plate and the frame is imitated with bolts, so the holes in the upper tube are also present. When creating the cylindrical fixtures it is important not to miss any surfaces, for example the insert parts have a separate surface that needs to be selected, as well as the frame has two walls which are in contact with the bolt. This gives in the end 4 surfaces on the tube, 2 surfaces for the insets, 3 holes in the index plate and one in the support block. The figure below represents the front and back view of those areas and a close up of the hole which connects the frame with the index plate.

The last step from the simulation of the bolts is again reference geometry. The first part is for the heads on the index plate as shown on figure number. With yellow are marked the surfaces for fixture and with pink the surface that determines the direction. The green arrows show the direction of restriction, in the case normal to the pink surface with displacement of 0 mm. The same fixture is done to the back of the tube holes and the support block.
Now it is time for the force application. The appropriate plane is chosen and gravity is applied. In this scenario it is very important to apply gravity since the frame is supposed to withstand the weight of all the possible equipment and components that are part of the test stand. Since including all parts in the assembly will extend and complicate the simulation too much, their weight is applied using force acting on the mounting table. As shown on figure 3.30 the force is acting over the whole area of the plate, because we are not interested in the connections between the plate and the other parts. The force will be distributed in the frame according to the connections/mates made between it and the plate. For these reasons and because the stiffness of the mounting is not important here, it is set to be rigid. After approximate calculation of the weight of all the components attached on top a result of around 500N was estimated. For safety reason and to allow usage of additional parts, if needed for future evaluations, the force applied on the plate as shown on the figure below is 600 N. The red arrow represents the direction of the gravity.

![Figure 3.30 Force acting over the mounting plate](image)

It should also be considered that the drill machine and the drill stand weights will influence the behavior of the frame. Again not to complicate the simulation, those components won’t be included in the assembly. The force will be applied through the index plate as shown on figure 3.31. This will allow the weight to be transferred through the connection fixtures made previously. The weight of the drill machine is 8.2 kg and the weight of the stand is 13.3 kg, which gives a total of 21.5 kg. After simple calculations this result is converted in to Newton. The value of the force is set to be 211 N. Even though the mass will not act over the plate as assigned in this simulation and most of the weight will be transferred to the support block, this is considered as a precaution act that assures that the stiffness of the those three tubes that are in contact with the plate is sufficient.
Last but not least, the torque is applied. It has a magnitude equal to 200N which is again the re-action force caused by drilling. The axis for direction coincides with the position of the connecting bolt. What is interesting is that even if positions of the drill stand is changed, the stress will not differ in size but only in location. In this case the position of the stand applies force localized in the right pipe, and if new simulation is created with changed position and new localization of force around the left pipe, the results will be the same since they have identical size and characteristics.
Before running the simulation an appropriate mesh should be created as shown on the figure below. In orange are the parts that are set to be rigid. As it can be seen the mesh isn’t too small which will lead to very long simulation. In the same time the elements are small enough to present the needed results in a good way. For the rigid bodies a bigger mesh is applied since their deformation isn’t calculated.

![Mesh of the frame simulation](image1)

**Figure 3.33 Mesh of the frame simulation**

The stress results are shown on figure 3.34. Since the biggest force in the simulation is 600N applied on top of the mounting plate, the consequences of it have the highest values. The picture on the left represent the automatic scale, generated by the program and the one on the right is in true scale. The results are in MPa and it can be noticed that they are relatively small and the deformations are insignificant. The maximum and minimum stress values are also indicated in the figure. On the right picture with red is highlighted the concentration of stress in the frame.

![Stress results of frame simulation](image2)

**Figure 3.34 Stress results of frame simulation**
Below in figure 3.35 is showed a close up of some of the concentrated stress areas. The problematic areas are caused by the weight of the components over the supporting tubes and the connections between them. However the magnitude does not require improvements or change of the parts characteristics. The maximum value shown on figure 3.34 is 4.897 MPa. The conclusion is that the structure is stable enough to withstand all the forces applied to it because the yields strength of the material is 235 MPa. This means that even if the drilling force is doubled or even 30 times more the frame won’t deform severely.

Figure 3.35 Concentrated stress areas in the frame simulation

Figure 3.36 shows the displacement of the different components in the assembly. The results are in mm and the biggest displacement is 8.570e-003 which means that it is insufficient to be considered as a critical or problematic.

Figure 3.36 Displacement results from frame simulation
Figure 3.37 shows the distribution of factor of safety. The scale is set from 0 to 20 and it can be seen on the picture that the construction is strong enough to withstand all the forces and masses applied to it. The results lead to the conclusion that material can be removed without any chance of fatal deformations. For safety reasons and because further evaluations may require different equipment or addition of other components, the characteristics of the frame will remain the same.

![Figure 3.37 FOS in frame simulation](image)

As explained previously the influence of the position of the drill stand is not sufficient to require separate study case. What is important is the position of the index plate with respect to the frame. As it moves up, the contact area between those two components is decreasing, which may lead to smaller area on which the force of drilling is spread. The consequence could be bigger displacement and higher stress values. Even though the previous study case lead to a conclusion that the frame is secure enough, for better understanding of its behavior one more simulation is conducted. In this second scenario the position of the index plate is the highest possible. The position of the holes on it and the surface created by split line around them is changed. Also after taking measurements from the original test stand model, the positions of the holes used for connection with the support block are changed. The previous study case is duplicated with the improvements made on the parts, which means that all the fixtures, forces and contact sets are identical. The new model is shown on figure 3.38.
Figure 3.38 Second scenario model for frame simulation

Figure 3.39 shows all the fixtures, forces and gravity applied on the assembly. The magnitude of each restriction is the same. This includes a torque acting on the wide split area with size 200 N, a force acting on the top surface of the index plate again 200 N, a force with size 600 N acting on the mounting plate and gravity force. The last thing that needs to be adjusted is the position of the axis which should coincide with the new hole position.

Figure 3.39 Force and fixture application on the new scenario
Figure 3.40 shows the results of stress and displacement for the second scenario. What can be noticed is that the stress is slightly bigger in its highest value. The reason for having maximum stress of 7.5 MPa is that the torque applied on the index plate is concentrated on smaller area. Since its value is the same the stress located in the area of the frame that is between the index plate and the mounting table is bigger. Still its value is significantly smaller than the yield strength of this type of steel which is 235 MPa. The biggest stress value is located in the same area as in scenario one. On the other hand the displacement is also a bit enlarged caused by the same reasons. To compare the result here is 9.041e-003 mm and in the previous case it was 8.570e-003 mm.

The last step of the static study for the frame is the factor of safety for this scenario. Again the value is above 20 for all the components of the frame. Figure 3.41 shows the graphic of the FOS. The conclusion is that for every position of the index plate or the drill stand, the masses and forces acting on the frame structure won’t lead to severe deformation or fatal damage. What is more interesting is that the element is constructed in a way that even if the force of drilling needs to be changed with respect to the evaluation in progress, the overall stability, safety and endurance won’t be changed. This also means that if the drilling machine needs to be changed with heavier one, or the blocks need to be replaced with bigger ones or even if more parts are included in the assembly, the frame will withstand the stress and pressure.
3.3 Simulation of Test stand

The last study is oriented to the overall behavior of the test stand. For this simulation a new assembly is created. It consists of the following components: frame, inserts in the frame, frame walls, support plate, index plate, drill stand, mounting plate, weight support walls, small frame, linear guides, positioning plate, support blocks, screws, overhead supports and the concrete block. Each component is as simplified as possible which means that all the holes are not present because they will overburden the simulation. The goal of this simulation is to investigate the overall behavior of the assembly. This process will be divided into two parts because the force of drilling is acting on two different surfaces. The first scenario will be the same as in the frame testing, where the force is applied by torque acting on the drill stand. The second scenario will consist of the same assembly but this time the force will be applied on a circular surface representing the position of one of the holes. In this way the influence of the force applied through the drilling tool can be explored. For more accurate results the simulation will be done for two different holes positions. Since the force can’t be precisely divided between the two surfaces, in both scenarios the total force of 200N will be applied.

Figure 3.42 shows the SolidWorks model that is used for the first scenario. As mentioned before all the components are simplified as much as possible. This means that they have the same size and material but all the holes are removed. This is done because there are too many
components and meshing and running the simulation will be a very time consuming process and can also lead to some errors. Also the result of interest is the general stability of the test stand and the severe deformations that can occur.

![Test stand simulation model](image)

**Figure 3.42 Test stand simulation model**

The first step of the simulation is to assign fixtures and acting forces. Shown in yellow on figure 3.43 are the fixed tubes of the frame. Their movement is restricted in every direction. This will lead to a fixation of the assembly in general, since all the other components are attached to them by the global contact set. The big red arrow represents the gravity force which is very important for this simulation because the weight of each component has influence over the stability. Then the torque is applied over the drill stand with magnitude of 200N. As in the previous cases the shift of the force direction is realized with respect to an axis coinciding with the position of the connecting bolt. And since the deformation of the drill stand is not important and it is used only to simulate the force of drilling acting on the assembly, it is defined to be rigid body. All the other components will remain deformable and they displacement will be analyzed.
What is interesting in this simulation is that after conducting it for the first time a conclusion was drawn that the function “mate” used several times to assemble the screws and the overhead support of the concrete are not sufficient to prevent penetration. Because the holes of the supports are bigger than the screws, the mate that is used to locate them is “Concentric”. This means that their centers coincide, but since there is no physical contact between the components the global contact set doesn’t prevent penetration. For this reason several contact sets are manually created. Figure 3.44 shows the two groups of sets with figures of the components that are part of them. In this way there won’t be any penetration of material between the screws and the block and between the screws and the holes of the upper supports.
The last step before running the simulation is to create the mesh. What is different here is that there are many components that vary in size and shape. For this reason a mesh control should be applied, which means that different parts are assigned with different mesh size. The smaller parts need smaller mesh so this is the first group. It includes the support system of the concrete and namely – the four support blocks, the four screws and the two overhead supports. The last two components that are included in this control are the two inserts in the frame tube. Considering the sizes of the parts a mesh with element size of 20 mm is assigned.

Now it is time to explain why the linear guides are not part of this group. Their 3D model is also simplified by removing all the holes but the geometry still remains complex. The presence of small edges and circular shapes such as those shown in red on figure 3.55, require finer mesh. For this reason the four carriages and the two rails of the linear guides are defined with element size of the mesh 2mm.

![Figure 3.45 Simplified model of the linear guides](image)

The next control is applied over the biggest components which include the five walls. In this group is added the drill stand since it’s defined as rigid and a fine mesh is not needed. What is important for these components is to understand the overall behavior with respect to the parts they are attached to. Since the biggest stress will fall over the frame the bigger mesh of these parts is reducing the run time without severe influence over the results. For this reason the element size is chosen to be 50 mm.

The other group with rather large mesh consists of all the tubes from the frame. What is different here is that those components need to be evaluated a little bit more accurate. Their deformation will be explored in more details so finer mesh is needed, but since the frame was separately tested there is no need to complicate the simulation too much by applying too small element size. For this reason an appropriate size of 45mm is chosen.

After the mesh control is defined for four groups, the mesh can be created. The size that is applied on the general mesh refers to the rest of the parts. This includes the support plate, the index plate, the mounting plate, the two weight supporting walls, the small frame, the positioning plate and the concrete block. They are assigned with mesh of 25 mm with tolerance of 0.6. The
element size is small because the behavior of those components is important for the end conclusion.

Figure 3.46 shows the mesh after it is created. It can be noticed that its size is different in the different components.

Figure 3.46 Mesh of the test stand simulation

Figure 3.47 shows two close ups of different components with finer mesh.

Figure 3.47 Close up of the mesh
Figure 3.48 shows the stress measured in MPa. As expected the biggest amount of stress falls over the frame. The maximum value is also shown on the figure and it is around the tube where the index plate is connected. What can be observed here is that the presence of the walls removes some of the tension from the frame. The deformation of tubes connected to walls is sufficiently smaller compared to the previous simulations. The maximum value of stress in this case is 1.4 MPa and it is located in the zone of the tube connected to the index plate, which is caused by the acting torque and the weight of the other components. Since the tube is made from structural steel 1.0038(S235JRG2), it is known that the yield strength is 235 MPa. Comparing the results to this value it can be concluded that the stress is insufficient to cause any permanent or severe damage to the structure.

![Stress results of test stand simulation](image)

**Figure 3.48 Stress results of test stand simulation**

Figure 3.49 shows the displacement in mm and it is presented in True scale. To bring to mind this means that the scale on the side shows the amount of displacement and the figure represents the actual and visual displacement. So the area marked with red has the biggest displacement, but it is only 3.341e-003. A conclusion can be drawn that the assembly is stable enough and won’t deform during drilling process. Figure 3.50 shows the strain plot of the structure.
Figure 3.49 Displacement results of test stand simulation

Figure 3.50 Strain results of test stand simulation
Now let’s look at the second scenario. In this case the force of 200 N is applied over the surface of the concrete block. It is important to investigate the behavior of the test assembly not only with respect to the reaction force but to the actual drilling force acting on the block. Figure 3.51 shows the assembly that is used for this simulation. The difference here is that since no force will be applied over the index plate, the drill stand can be suppressed, which will make the process faster. The next step is to create a small split face on the surface of the block where the tool will be in contact. The size of the circle is 20mm which corresponds to the diameter of the drilling tool. For more accurate results the simulation will be conducted two times with two different holes positions. Because all the 16 positions are located on the same component and the applied force is the same, there is no need to test all of them. For this reason the top left and the bottom right holes are chosen for the two scenarios. This covers the most extreme possibilities and will give results for the worst possible scenarios.

In order to prevent the need of creating the contact sets from the previous case, the first study is duplicated. This means that every force, fixture, contact set or mesh control is copied to a new study. This saves time, because the only thing that needs to be changed is the force that is applied. When the study is created the program gives a warning that some of the surfaces can’t be located. This is because there is a torque applied to a component that is now suppressed. After deleting it, a new force with magnitude 200 N is applied over the concrete as shown on figure 3.52. The last step before running the simulation is to remove the drill stand from the mesh control and create the mesh with the same element size for all the other parts.
Figure 3.52 Forces applied in second scenario

The generated result for stress values is shown on figure 3.53. The maximal value is 1.8 MPa which is remarkably smaller than the yield strength of all the materials used in the assembly. What is different here is that there is no sufficient stress around the index plate which could cause deformations. It is only concentrated over the mounting plate which leads to results similar to those of the frame simulation. What is interesting is to see that the stress on the surface where the force is applied is not the maximum value. This is because the force is translated to the other components but each part is adding additional pressure caused by its own weight. For this reason in the detailed testing of the frame the load applied on top of the mounting plate was more than the actual weight of the components, providing results for a scenario where additional equipment is added.

Figure 3.53 Stress results for second scenario
On the close up shown on the figure below can be seen the stress that is distributed through the tubes reaching the bottom of the frame. Even though its magnitude doesn’t surpass 0.6 MPa it was considered during the decisions made for the design of the frame.

![Figure 3.54 Close up of the stress results in the bottom of the test stand](image)

Figure 3.54 Close up of the stress results in the bottom of the test stand

Figure 3.55 shows the results for the displacement in True scale. The highest value is 4.954e-003, so the conclusion is that there are no severe deformations caused by drilling. The displacement is realized in two directions. One is result from the acting force on the concrete causing the assembly to shift to the back and the other is effect of the mass of the components – pushing the assembly down.

![Figure 3.55 Displacement results for scenario two](image)

Figure 3.55 Displacement results for scenario two
Figure 3.56 shows the strain distribution in the structure. As seen in the close up below it is located in the area where the block is pushed by the drilling force in to the support blocks and the contact surface between the mounting plate and the frame.

![Figure 3.56 Strain results for scenario two](image)

Now these results can be compared with the next case, where the force is applied on the bottom right surface. Figure 3.57 shows the position of the split surface which again has the size of the tool tip. The same force of 200N is applied and also the assembly has the same fixtures and contact sets. The aim of this scenario is to investigate if the drilling is conducted in the lowest part of the concrete, will have different influence over the overall behavior of the test stand.

![Figure 3.57 Second position where force is applied](image)
After the simulation is done the results are analyzed. The stress is shown on figure 3.58, the displacement on figure 3.59 and the strain on figure 3.60. The first thing that can be noticed is that the stress results are the same as in the previous scenario. The maximum value is 1.4 MPa and its location is the same. This leads to a conclusion that the position where the drilling process is conducted doesn’t influence the overall behavior of the test stand. No further simulations of the other 14 positions will be needed. On the close up in the figure of the stress can be seen that the value acting over the tubes connected to the mounting plate is only around 0.8 MPa, which means that there is no need to improve the frame by changing its characteristics. When comparing the displacement results it can be seen that the maximum value in the second scenario is smaller than in the first, respectively 3.725e-003 and 4.954e-003. The reason is that the drilling is done closer to the wide are of the positioning plate and this leads to a bit more stable environment because the support is stronger. On the other hand the strain is a bit higher - 1.622e-005 compared to 2.369e-005.

Figure 3.58 Stress results for second hole location

Figure 3.59 displacement results for second hole location
What is important to be noted is that the structure behaves well and the presence of enough tubes provides very stable basis for different process evaluations. The diagonal tubes in the front and the welded walls bring additional firmness as intended. The results of all the simulations show that the design of the test stand and the choice of materials were accurate and they lead to good outcomes. The magnitude of the results assures that even if larger forces are applied over the structure it will maintain its resistance.

### 3.4 The assembled test stand

After each was designed and tested it is safe to proceed to construction. A detailed drawing is created for each part separately and sends for production. The components are then welded and connected together. The end result can be seen on the figures below.
Figure 3.62 Test stand from the back

Figure 3.63 Test stand front view
Figure 3.64 Close up of linear guides

Figure 3.65 Mounted concrete block
The goal of this master thesis is to create a system for collaboration with Diamond Coring Tool DD 150-U and Drilling Stand DD-ST 150-U CLT. It consists of a test stand that is designed to withstand a great amount of drilling processes with the same stable conditions and good quality of the holes. Drilling is produced in 20x20x20 mm concrete blocks with constant force. The first step is to create a 3D model in SolidWorks 2013 of the stand and choose appropriate size and material for each component. The design allows easy change of components, application of constant force, drilling without influence of a worker, usage of encoder EC82-125-5 and easy relocation of the tool with respect to the 16 holes that need to be drilled on one block. A new system of movement is made in order to provide constant drilling force throughout the whole process.

Before all the parts can be ordered for production, the test stand is evaluated using FEM simulations. In this way the efficiency of each critical component where a lot of pressure or force is applied can be tested. Also the overall stability of the system is important and to save time and money for creating prototypes or rebuilding parts their displacements, stresses and strains are numerically calculated. After analyzing the results, a conclusion is drawn that the yield strength of the materials used in this assembly are significantly larger than the maximum stress concentration caused by drilling forces or mass distribution. This leads to negligible displacements and strains, proving that the test stand and its parts are stable enough even if additional equipment is added or the force applied from drilling is enlarged more than two times.

All these different aspects prove that the test stand that is designed in this master thesis is stable enough to carry out severe number of drilling processes. This allows investigation of the different aspects of drilling in concrete and analysis for possible improvements in the process. Each part is constructed and connected in a way that allows easy replacement or change which gives the possibility for a great amount of diverse tests and evaluations. The stability of the test stand also gives perspective for additional equipment if needed or even change of the material under investigation.
Bibliography