SolidWorks Bottom-Up versus Top-Down Approaches.

The Capacity of the Top-Down Modelling.

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ABSTRACT

Due to the significant importance of the 3D modelling technology for the engineering companies, the two types of modelling approaches which determine the work process of the were examined.

In the following study, a comparison of the design intents and accurate evaluation of the strengths and weaknesses of the Top-Down modelling approach for the utilisation in the 3D CAD software – SolidWorks - were performed in order to identify the capacity of the method in the company's environment.

In order to obtain reliable evidence, the collaboration with a real company was arranged, the collection of the theoretic knowledge and the modelling of a shipyard box for obtaining solid empirical data were performed.

The Top-Down approach which was analysed in comparison to the Bottom-Up intent was found to be less stable, more complicated and time-consuming. When considering the results of the study, the Top-Down philosophy was considered as a not satisfying method for the entire realization, however, its tools could be solely implemented to reach beneficial outcomes.

The outputs of the thesis have an implication for the cooperating company as it will influence the development of the standard procedures' strategy to enhance the capacity of the organisation.

Keywords Design Intent, Top-Down, Bottom-Up, SolidWorks, Parametric Modelling

Pages 48 pages

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1 INTRODUCTION

Do people ever ponder about the creation methods of objects which surround them? In past centuries prototyping was the only existing solution. Manufacturers were to produce real items with theoretically estimated parameters to face numerous errors and shortages in the design. Consequently, a long period of time was allocated for the conceptualising stage of product development. With the development of the computing devices which were capable of performing complicated arithmetical calculations and storing the result following the prewritten instructions. The computer becomes an invention which has introduced a new era in product development. The engineering world has changed dramatically since the release of first computer-aided design systems which were capable of creating a two-dimensional drawing with the help of primitive sketching tools, such as line, point, circle and other. The evolution of the programmes to the three-dimensional led to the development of the new field of the industry: 3D CAD modelling. The software offers a high range of tools which allows building volumetric objects by creating and combining standard geometries. Contemporary programmes simplified the procedure significantly empowering ordinary people to design objects without engineering education. However, when lacking the basic knowledge about the manufacturing process, people might not consider the types of relationships which exist in-between the components of the model and production's feasibility.

In regards to the increasing importance of 3D modelling, the proper understanding of the term "design intent" is significant. It presumes the existence of several approaches to the modelling process which defines the procedure and principles of an object's creation. The selection of a suitable method directly impacts the final outcome.

The aim of the research project was to determine characteristics of Top-Down and Bottom-Up modelling approaches, reveal their advantages and disadvantages, conduct a practical case by modelling a product with two design intents and evaluate the capacity of the Top-Down for the utilisation in the realistic conditions of the functioning company.

The study was completed in the collaboration with the engineering company who has proposed the research problems. The name of the organisation was excluded from the text due to the Non-Disclosure agreement. In the following chapters, it would be referred to as a "third party" or "Company". The obtained information will be used for the further studying of the topic and estimation of the profitability of new techniques' implementation in the common project's workflow in the future. Additionally, the Company, being a follower of the lean paradigms related to the continuous improvement, might potentially test revealed tactics as an alternative solution for the upcoming projects.

Due to the specificity and relatively low number of reliable and structured sources regarding the researched topic, the project's objective was to collect distributed information concerning the studied questions and present it in a logical and readable form to provide quick access to the structured data. The study was divided into seven chapters. Design intent's type would be explained in Chapter 2 to give an overview of the approaches. Then, the CAD software and concrete Top-Down modelling technique would be reviewed. Chapter 5 outlines the empirically conducted study case which was granted by the third party. Lastly, the capacity of the Top-Down approach would be estimated in Chapter 6 summarising the collected data.

2 APPROACHES IN 3D CAD MODELLING

The controllability is an essential factor during the 3D CAD modelling. In order to design a reliable component, the way how it is going to be done should be forethought. The term "design intent" emerge from this approach. It describes what kind of relationships exists between elements and how it determines its alteration. More specifically, the change of parameters in one place could affect other parts regarding the design intent; it is important to be aware of those links and the way how they impact each other. To have control over the component, an effort to structuring, planning and implementing should be well-thought-out at the early stages of design considerations. With this in mind, it worth spending the time before work, to receive a great output at the end. (Martin, 2018)

Due to the difference in people's attitude to the modelling and common intention to be consistent and receive a maximum profit out of the workflow, several approaches have been developed. To look from the general perspective, there are three levels of modelling where these methods could differ. To begin with, all parts start from the dimensioning scheme or sketch where the main parameters are stated. Depending on the way, how it is done, the part would endure changes in a different way. Figure 1 illustrates the point. The location of the holes was stated with undefined parameters which have certain references between each other and, as well, referred to the edges of the body. Under this circumstance, they have kept the specified proportions, even though, the body's size was changed. If otherwise, holes were defined with exact fixed values, they would stay at the same distance in relation to the edges. (Martin, 2018)

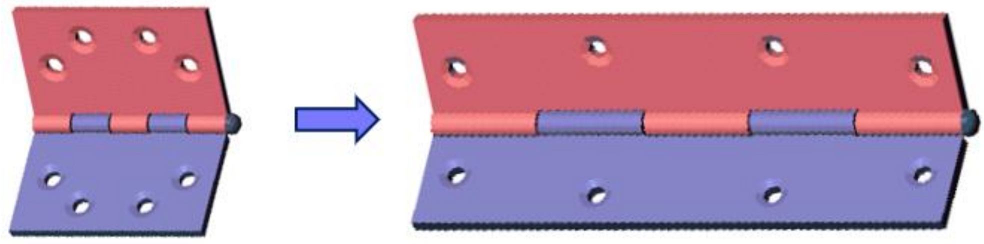

Figure 1. Example of the design intent. Sketch level (SolidWorks, 2012)

Another level of the design intent influence is a component modelling. Apart from the proportions, the dimensioning of the features, such as extrude, revolve, swept and other, is crucial for the following work process. To support this point, an example case could be observed in Figure 2. The central hole of the part was supposed to go all through the part but was dimensioned to be straight five millimetres provided that the length of the part was initially five millimetres. Under these circumstances, as soon as this parameter of the part has changed, the hole could not fulfil the requirements anymore as it was no longer coming through the body. (Martin, 2018)

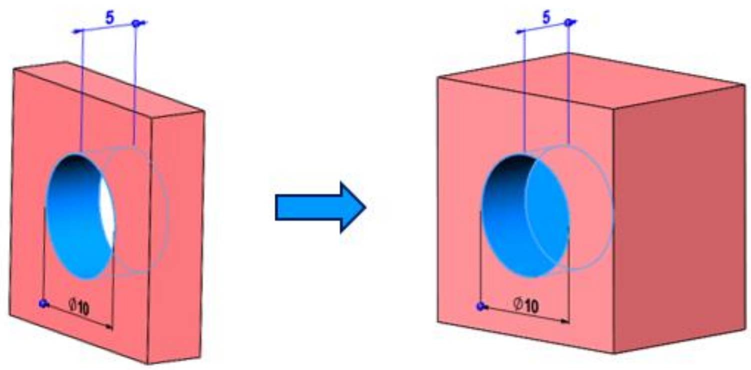

Figure 2. Example of the design intent. Part level

Additionally, the design intent may vary at the assembly level in a similar manner. The output of the modelling process will fully depend on the way how elements were linked to each other.

After the selection which type of design intent suits most to the current project, it should be followed all along the modelling process to keep everything structured and organised. Otherwise, switches might result in numerous unfixable mistakes the worst of which could be an inability to use the part.

During the development of computer-aided design procedures, several approaches to the design intent have been formed. They are going to be described in the next chapters.

2.1 **Bottom-Up**

The most traditional and, consequently, the most common way of 3D modelling is a Bottom-Up approach. It is widely used because of its simplicity. As statics states, 80 per cent of the 3D CAD models are created Bottom-Up, proving it to be the most favourable technique among engineers and designers. The reason for it is that most of the engineers were taught 3D modelling in this way provided that the workflow of a Bottom-Up assembly better mimics a process of real manufacturing what is more logical for the human mind. The world has developed from elementary to complex. Likewise, a Bottom-Up approach tends to work: individual elements are designed and specified separately and then united together, placed, preferably, in the way it would be done in real life. Figure 3 shows the main principles of the Bottom-Up method.

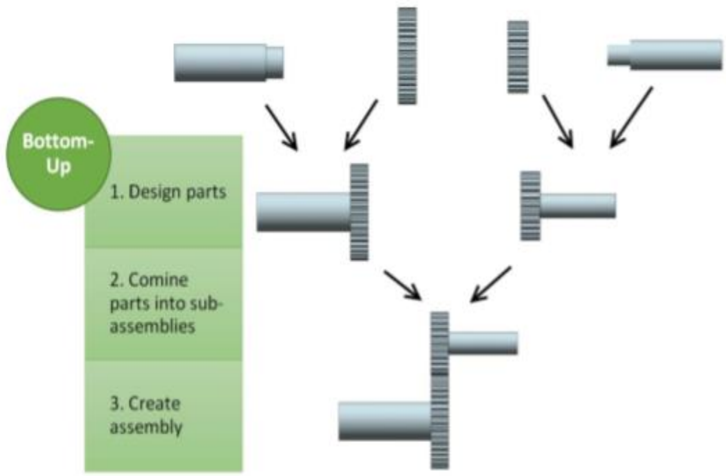

Figure 3. Steps of the Bottom-Up modelling (Juustovaara, 2015, p. 15)

This method could be successfully applied when modelling standard parts or redesigning assemblies. The point of focus, in this case, is on each individual component and how parts are fixed together to form a structure. It is a simple method as no external links exist which make one part depend on another. It easily breaks and can destroy parts provided that some incorrect actions are performed. All components are self-depended and have constraints just to piece them all together in assembly.

2.2 **Top-Down**

Top-Down philosophy is a method of planning and designing models starting from the main assembly and then decomposing it into minor parts. The overall process is, typically, done within one central part that drives all critical parameters of the whole system. This approach is also called stepwise or in-context modelling and considered to work at a "macro" level, from the general view to specific. Firstly, assessment of the vital functions of the concept is performed and, afterwards, construction of necessary geometry, features, surfaces and solids inside the dominant assembly and in reference to it is done.

The Top-Down method requires an accurate planning and a certain level of awareness from the designer before the actual modelling would start: the central structure should be created, preferably, all interrelations should be thought out, the step-by-step workflow should be defined in order to avoid unsolvable repercussions at the beginning. As Paul Munfould (2013) has stated in one of his public seminars: "Before you can model your design, you must design your model". This attitude has a positive impact on the result of the project conducted with the help of in-context technology as it highlights the necessity of the proper preparation and mind work before the start of the work.

The whole assembly is controlled with a rigid foundation which determines the dimensional data of other elements. In order to manage all the internal elements, the driving entities should contain a so-called skeleton, as one decent way of manipulating the file. The skeleton is made in the form of a sketch which represents all main characteristics of the part, including size, volume, location in the space, possible mechanisms. Figure 4 illustrates the simplified workflow of Top-Down designing. As it could be seen, firstly, the sketch of the final design of the assembled part was drawn, secondly, it was separated into two sub-sketches creating a parent-child relationship and, lastly, four components were modelled in accordance to the sketches of a higher level.

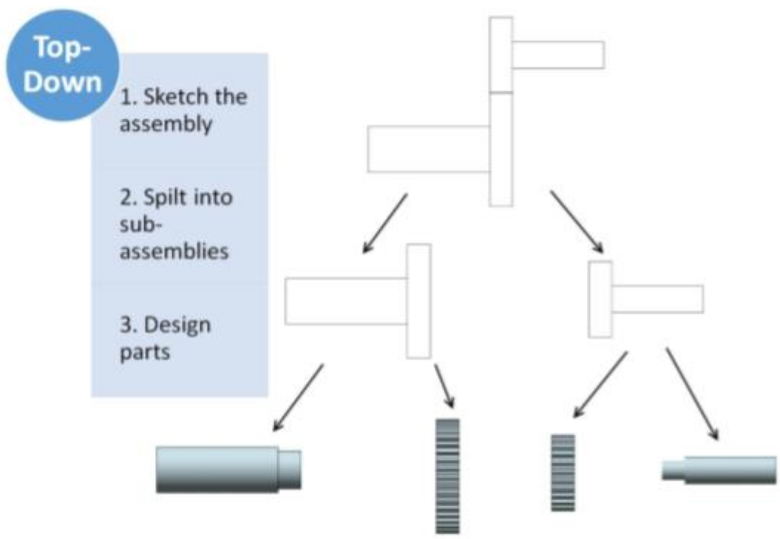

Figure 4. Steps of the Top-Down modelling (Juustovaara, 2015, p. 16)

This methodology is an appropriate option whenever the modelled system consists of a large number of sub-assemblies and should withstand a big number of modifications before it is going to be finalised in spite the fact it is rather interactive. Moreover, it facilitates the changes as all the adjustments could be made inside of one file without a need to work with each element independently. Whether something is altered or deleted, the overall structure will not fail or disintegrate because parts are mated to the skeleton.

2.3 Middle-Out

Middle-Out is a mixture of the Bottom-Up and Top-Down approaches that has absorbed both positive features but, simultaneously, several drawbacks, from each of the methods and evolved into the independent philosophy bringing its techniques in the modelling process. When assessing it from the human perspective, this method could be referred to the regular design: before starting modelling, people typically plan the process, make some calculations or estimation, conceptualise. Eventually, the creation of the model starts. However, not all parts are going to be modelled – some could be exported from the library or from previous models.

When the assembly step comes, even though several parts that have already been modelled in advance, could be inserted and pieced together, it is possible to develop others in the context of the assembly environment attaching them with external references to other elements. The simplified diagram could be observed in Figure 5.

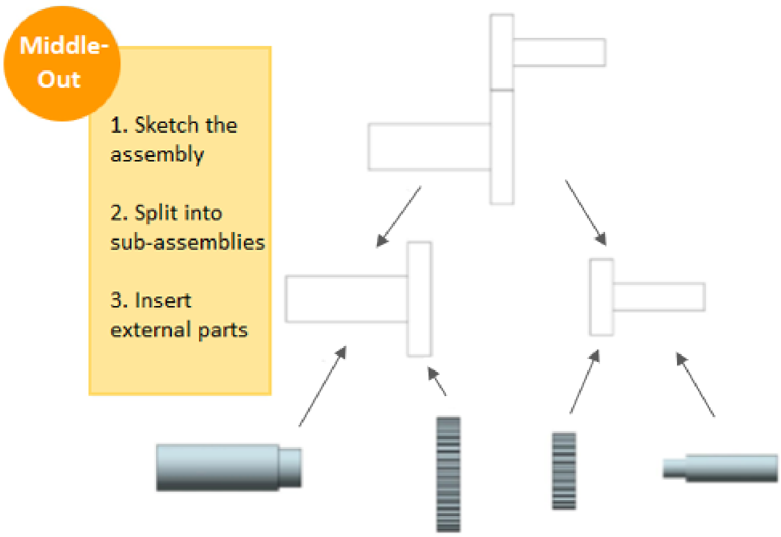

Figure 5. Steps of the Middle-Out modelling. Modification of the (Juustovaara, 2015, p. 16) scheme

The Middle-Out approach could be a helpful tool on the condition of being implemented into a design process in the middle of the project when an already existing structure should be altered and improved. The case might come from another customer who has already contributed some time and money into the project. Consequently, there are no resources to start from the beginning meaning the project should be continued from the place where the work was interrupted. At this point, it is necessary to work inside of the assembly which was already done but some improvements could be done both in-context and with the insertion of the necessary external elements.

3 3D CAD SOFTWARE

Nowadays most of the objects that exist in our world are modelled with a three-dimensional computer-aided design. It performs numerous mathematical and geometrical calculations which are hidden behind the process but results in the ability to model objects in virtual reality starting from two-dimensional sketch and then transforming them into a volumetric part. Special tools have been developed for this purpose allowing, for instance, to extrude, revolve, scale or pattern elements. Working with software which specialises in the 3D modelling, designers are capable of making products to have similar weight, size, density, physical properties and appearance to a real object, even though, it does not exist. Due to its realistic parameters, it could be inspected and tested for its withstanding properties what is demanding for the product creation process.

Numerous software has been created to satisfy various users' requirements. For instance, such programmes as SolidEdge, Catia, NX, Fusion360, Tinkercad could be listed. In the following paragraphs, three professional 3D solid modelling software is going to be reviewed to show their capabilities and reveal unique tools that could be helpful in different circumstances. Even though the description of programmes will be presented, it is thought to be an opinion-based revision which is determined by user experience. Overall, most of the high-end 3D software offers a similar set of tools and features and identic products could be designed within each of them.

3.1 SolidWorks

SolidWorks is a common worldwide used CAD and CAE software that is especially known for its excellent adaptability for numerous purposes in most of the industries, automation, user-friendly interface, logic and easy learning process as key attributes. It utilises a parametric based approach and has a process tree to follow the history of the modifications of the part.

All three-dimensional parts start from the 2D sketch where basic geometry is drawn and then converted to real objects. The sketching tool is outstandingly intuitive and easy to work with since it offers a wide range of helpful features. Although SolidWorks could be manipulated in a conventional manner, it could be also operated both with keyboard shortcuts - a combination of certain symbols that call out a command - and mouse gestures what is highly appreciated by the designers who tend to customize the program for their needs. A quick window selection is, similarly, a helpful tool that arises as soon as some features are applied offering instant access to the possibly required functions. It appears near the cursor what introduces SolidWorks “heads-up” policy which is intended to help users stay focused on the working area with no need for searching tools from the feature bar.

Editing of features is simple as it could be done either by dragging an element and receiving instant visual feedback from the graphics area or numerically manipulating values. Some of the features that typically could be applied just in one manner are adjustable in the SolidWorks. Chamfer and fillets could be asymmetrical, gradually increasing or decreasing the size across the edge which could be selected fully or with segments according to the geometry. In regards to adaptive techniques, one sketch could be utilised several times to create different features and geometries, for example, removing or extruding body.

Hole wizard is a tool that controls actions concerning holes. It includes several types of holes that are used during the manufacturing: countersink, counterbore, through holes, tap holes and dowel holes. Moreover, due to the large and comprehensive database, hole wizard is capable of working with standards and concrete hole sizes that could be selected from the library. The software calculates and creates the geometry automatically as soon as parameters are assigned. Consequently, SolidWorks assures that modelled holes fit classifications and meet standards.

To handle technical documentation, SolidWorks offers a mode for creating a drawing from the part. All part’s orientations are presented in the pallet where an appropriate view could be selected and then pulled to the working area. When placing the first orientation of the model, the software automatically suggests projected views including an isometric one. Tolerances could be set while the creation of a dimension allowing to fill all information without applying other functions. If to consider the case that some of the geometries do not assure the requirements what is noticed at the stage of preparing a technical drawing, the dimensions could be changed from the drawing mode what would propagate to the part accordingly. Even though an automated dimensioning and balloons placing could be used to reduce the time, the function is not accurate and could repeat similar elements what is hardly noticed in the complicated drawings.

Part’s assembly being one of the consequent steps of the designing process, is thought out well in the SolidWorks providing users with many convenient features. Firstly, although parts could be brought to the assembly with the "open" function, it is efficient to pull it to the graphics area from another window without searching for the right command. SolidWorks, moreover, predicts a most possible solution for the mating to mimic a realistic behaviour of the component and propose it for the user. The same technique works for the standard parts that are inserted from the database and which have a wide range of sizes and parameters that could be selected. What is more, the parameters of certain parts could be adjusted to fit the geometry within SolidWorks. To illustrate the point, the following example could be observed The washer was downloaded from the library and constrained to the pin. As soon as mating was performed, the washer has increased its size to fit the pin. To support the point of the convenience of the software assembly mode, it could be stated that parts could be copied with remaining their references and then added to the new location. When the surface to where it should be mated is selected, a placing is done automatically.

To conclude, SolidWorks could be used for the projects of different scale characterising it as a versatile software which is suitable for various types of customers. It is one of the worldwide leading programs that provides great control over the parametric modelling of parts. It was developed with an intention to empower users with a clear and efficient workflow what allows to concentrate more on the design and features than on the modelling process.

3.2 PTC Creo

PTC Creo is an integrated family of various CAD software that is created to fit for different tasks that engineers might need to accomplish. The division of the functions into several applications brings flexibility to the work process as each user can find appropriate software for concrete needs without purchasing additional but useless features.

PTC Creo is known to be robust for its learning curve as it is difficult to familiarise yourself with its interface and process workflow. Its multi-CAD data usage is convenient as it is capable of working with models which have different file´s types. Additionally, it can successfully perform complicated simulations which could be used to optimize the product on the designing stage.

Creo is designed to manage with extremely large assemblies and complex geometries and perform all actions with them faster than other programs. As it was tested by Tech-30 (2019), Creo has opened and enabled editing of the heavy assembly 2,5 times quicker than a SolidWorks. This is significant for the development of complex and highly-detailed products. What is more, the files that are created in Creo, are lighter than the similarly made models from other software. (Tech-30, 2019)

3.3 Autodesk Inventor

Autodesk Inventor is CAD software that has been released on 1999 and has gained its followers regarding its ability to work with complex assemblies, mesh modelling, large design reviews, hybrid modelling, ILogic and other tools that Autodesk has developed and included into the software.

Inventor empowers designers to have work done admittedly faster. It has internal “design accelerators” which helps to iterate models on the early stages what results in an efficient and thoughtful design. The optimization of parts is essential for the companies as any changes are ten times more expensive when proceeding forward during the product development starting from the concept step and ending at the prototyping.

Autodesk Inventor includes both a basic toolset that is used for the modelling and many reliable features that help to reduce repetitive tasks. It can create various shapes or geometries allowing to model any idea into a real part due to the support of the parametric, freeform and direct modelling modes.

Another positive aspect of the Inventor is its stability meaning there are fewer interruptions during the workflow. The powerful and flexible software is able to maintain a smooth modelling of large assemblies what is highly appreciated by designers.

Work with the same assembly could be challenging when several engineers are assigned to complete it together. Whenever any modifications are applied to the part, they will not propagate to the other copies of the same model. As elaborated above, simultaneous work is almost impossible. Icloud based reviews and Autodesk Vault product data management allows companies to follow the methodology of concurrent engineering and develop a model from different aspects. Additionally, Inventor can identify files from the external software and read any file format what makes an integrated work between partners and customers simpler and more straightforward.

4 TOP-DOWN MODELLING METHODS IN SOLIDWORKS

To control and structure the Top-Down modelling, different methods have been developed. Even though they are used with the help of different tools, the aim is still similar: produce a reliable and stable to alterations assembly with great control over it. Described techniques might have similar concepts in their background, but, eventually, they have a share of their own positive and negative aspects.

4.1 Skeleton

Skeleton or skeletal modelling as it is commonly called, is a technique which is upbuilt around an outline. To put it differently, a part or an assembly is controlled with a separate sketch that includes all sufficient geometries and shapes which will drive all the other elements in the system. An example of skeletal modelling is represented in Figure 6. The routing between the sketch and components is done. Consequently, the changes in the skeleton will automatically affect all the parts of the system.

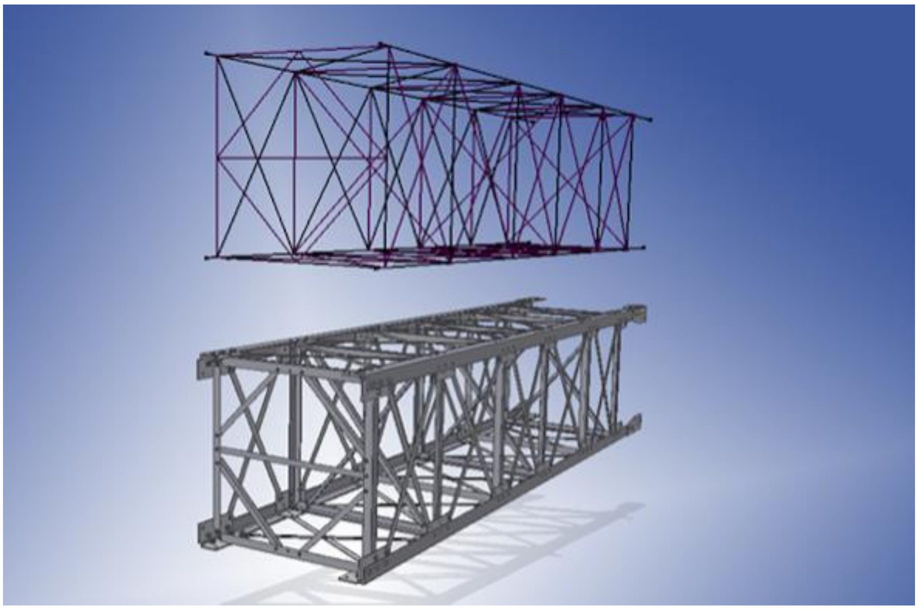

Figure 6. Skeletal modelling (Autodesk.Help, 2019)

There are several types of a skeleton that are generally used in the Top-Down:

- Location skeleton
- Space reservation skeleton
- Published skeleton
- Control skeleton
- Motion skeleton
- Design skeleton
- Shape skeleton
- Detailed skeleton
- Double skeleton

To start with, a skeleton should not be too detailed to work properly under the circumstance it would add undesired complexity and weight to the assembly. Secondly, external references should be accurately handled. To achieve this, the skeleton should be preferably modelled in a different file being grounded to the correct place or drawn before all other elements in the file. Mating of all elements to the skeleton is done on the condition that it is a central drive part which will control all modifications. Therefore, the linkage would result in the solidity of the system.

As soon as a skeleton model is finished, the referencing of subassemblies or individual elements could be started. This process results in the establishment of flexible dimensions that are not fixed to the concrete values but refer to the external master sketch that acts as a core element.

The drawback of the skeletal modelling lies in the fact that in order to modify the whole system, the original skeleton should be opened and changed. With this in mind, the initial designer is the only person who can rebuild and update the file if the document is not shared within, for example, a product data management system, resulting in inefficient usage of time.

4.2 Layout sketch

Layout sketch is a method that is also based on a sketch but the purpose of it is completely different from the skeleton modelling. The concept of the layout is to create a side-view three-dimensional sketch of the structure inside of the assembly relating all blocks - separately made two-dimensional sketches - so that the mechanism of the system could move as it would be in the final part. It is possible to test how the body will act when entities will be moved. Moreover, it could be as detailed as required. (Autodesk. Help, 2016)

Tentative outlines of all the features and geometries should be created and fixed correctly and then the physical blocks could be generated from the layout, keeping all elements flexible. When all the elements are modelled, the kinematics of the assembly will be maintained as at the sketch stage. In Figure 7, the layout sketch and the geometry that was modelled out of it are represented. (Autodesk. Help, 2016).

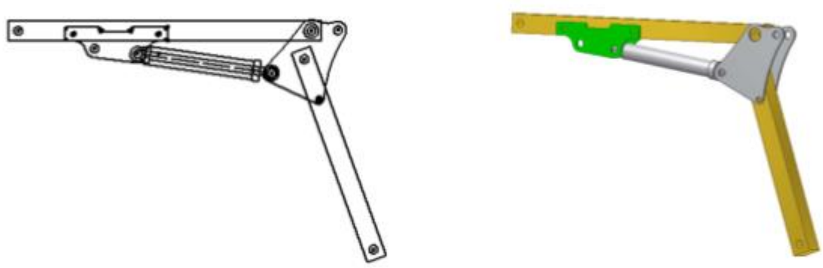

Figure 7. Layout sketch and derived geometry (O'Flaherty, n.d.)

Blocks are an essential part of the layout sketch. Block, as it was briefly mentioned above, is an independent sketch or a group of them that were created separately from the layout sketch and then inserted into it with a conversion to a block. When transforming the sketch into the block, it is important to take an insertion point into consideration as it allows to assign the position of the origin to the convenient place for the following design. Some tools, such as an ellipse, a slot, polygons, offsets and other, are disabled in the layout mode what explains the necessity for blocks' utilisation. However, when starting an individual sketch these functions could be accessed allowing to utilise them and next transform the sketch into a block to be inserted into the layout. (Bukovski, 2017)

When the layout is completed, the components could be made from the drawn geometry. Each block could be extracted individually allowing to model separate components. They are already placed into the space relative to each other but not necessarily mated due to the referencing at the layout level.

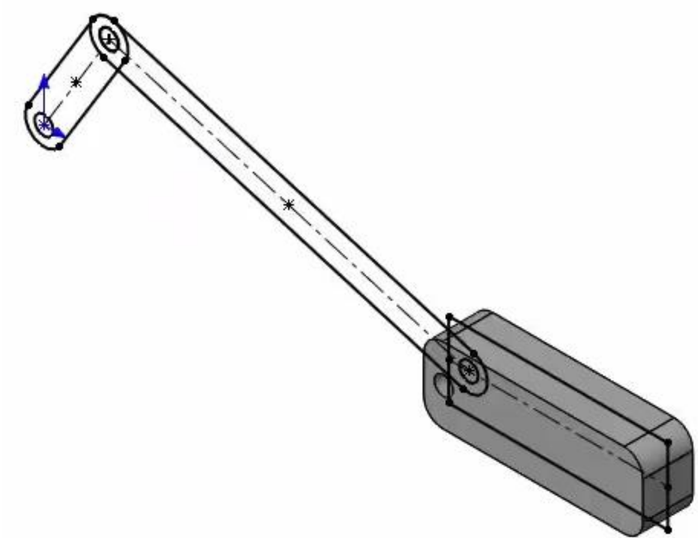

Figure 8. Layout blocks with extruded geometry (Bukovski, 2017)

4.3 **Adaptive parts**

The adaptiveness comes from the ability of the parts to change automatically due to the fact they are not fully constrained. When such a part is added to the assembly, the adaptive parameters could be linked to the core geometry what would result in simultaneous modification of the part if the changes are applied to the drive body. Unconstraint of the part is a key factor to make the part adaptive. An adaptive part could be inserted into numerous independent assemblies with different constraints to satisfy special needs resulting in reusability of the part. However, all the files that were exported from different CAD software are assumed to be not adaptive as they could not be modified and are supposed to be fully defined. (Autodesk.Help, 2019)

The adaptiveness could be set at almost any level of the modelling process:

- Sketch
- Feature
- Part
- Sub-assembly

For example, to make a sub-assembly adaptive, its parts and their features should be kept adaptive as well. However, it is impossible to bring adaptiveness to the assemblies as, obviously, it is a top level of the modelling and it does not refer to any other geometry. (Autodesk.Help, 2019)

The adaptive parts could be especially effectively utilised when the size of the final product is not clarified or when the parameters of one element depending on the configuration of another part. As an illustration of the adaptivity of the part, Figure 9 could be observed below.

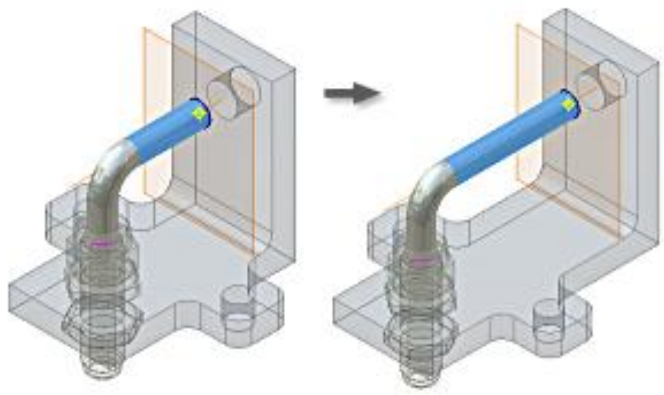

Figure 9. The adaptivity of the assembled parts (Autodesk.Help, 2019)

The parts should have linkages to external parts in order to be adaptive and functioning. However, external references are thought to be ambiguous as being a powerful tool, meanwhile, it adds heaviness to the model what makes the rebuilding complicated and breakdown easy.

4.4 **Flexible parts**

To prevent a misunderstanding, the term "adaptive" does not have the same meaning as "flexible" in case of CAD modelling. In most of the software, assemblies do not keep the ability to perform kinematics which was created at the part level. To make the movement act properly without a need to return to the lower-level part file, change the position of the adjustable element and rebuild the assembly, the component could be solved to have a flexible configuration instead of rigid. This parameter could be set in the component properties. Overall, flexibility is a state of the body that allows controlling the mechanism in a model.

The problem with the flexibility of the part arises when the part is modified by several engineers. As activation of flexibility is not stated in the main graphics and could not be noticed forthwith without an intention to figure it out, designers might accidentally break the whole system if they do not check the functionality of the part beforehand and continuously thoroughly operate with a model.

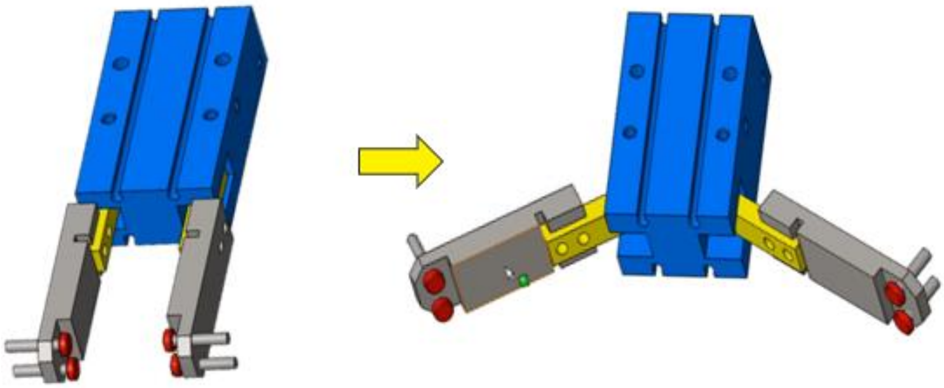

Figure 10. Flexible subassembly (Javelin Technologies Inc, 2013)

4.5 **Derived geometry**

Derived geometry aims to create an exact replication of the model that will maintain the linkage between two parts. The derivation could be held both at sketch, part and sub-assembly levels.

To create a derived geometry, the already-made sketch or part and the place to where it is going to be located should be selected and, afterwards, the “derive” function could be applied. The derived element will appear with an external linkage to the master part. The mating is a main benefit of the derivation as it automatically translates the modification of the initial part to the derived ones.

Derived sketch’s parameters cannot be redesigned separately from the original sketch because its geometry fully depends on the part from where it was derived. However, it could be not only positioned to various places but also related to other entities. Otherwise, if the derived sketch should become an independent unit from its parental geometry, the connection could be broken with “underive” function. Once the link is removed, it could not be restored. (Dassault Systemes, 2018)

The derive function is not similar to the copy/paste one. The difference is illustrated in Figure 11. The master sketch is represented with a yellow colour. When the sketch which is drawn in red colour is, firstly, copied and then pasted to the position, it does not have any bonds with the initial geometry so they could be changed independently with no impact one on another. On the contrary, the blue sketch was derived and then related to be located at the centre of the side wall. Its parameters could be altered only if the initial sketch will be modified.

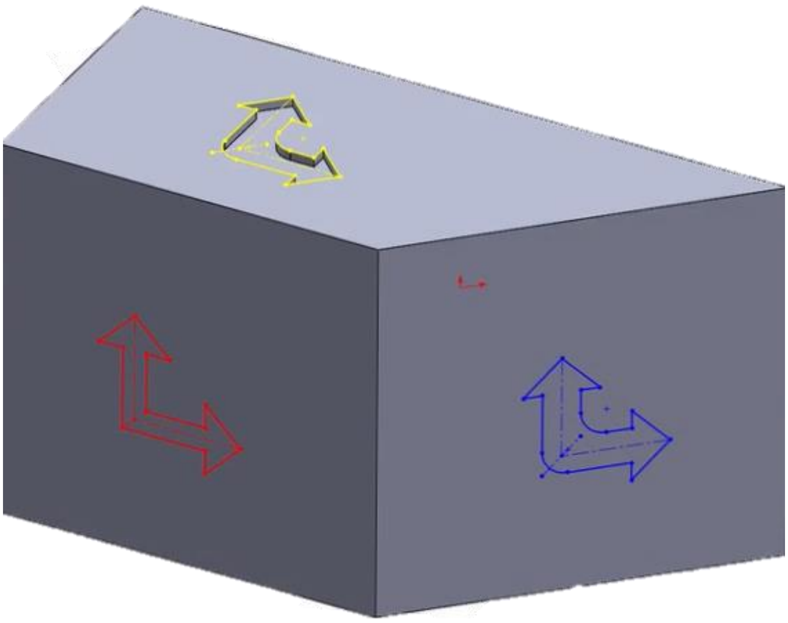

Figure 11. The difference between the derived and copy/pasted sketch (Grover, 2015)

The derivation increases the efficiency of the work as there is no need to change all similar elements if it is required. To put it differently, all entities are related to the main unit: the redesign of only one element will propagate to all derived parts. What is more, when several derived objects are inserted in the assembly, some of their features, that do not have an exact value, could be excluded from the transferring what will reduce the loading time and improve the performance. (TEDCF Publishing, 2013)

If to talk about another positive aspect of the derivation, the part could be placed on the correct position without any additional constraints that are easily broken when working with large assemblies. When a prime sketch is created and saved, it could be then opened distinctly within a derived feature. In the new file, elements could be modelled out of the sketch and saved independently. As soon as all units are created, they could be inserted in the assembly file and there will not be any need for the constraints if elements will be grounded during transferring what would locate them in the place where they were initially geometrically modelled. (MGFX, 2015)

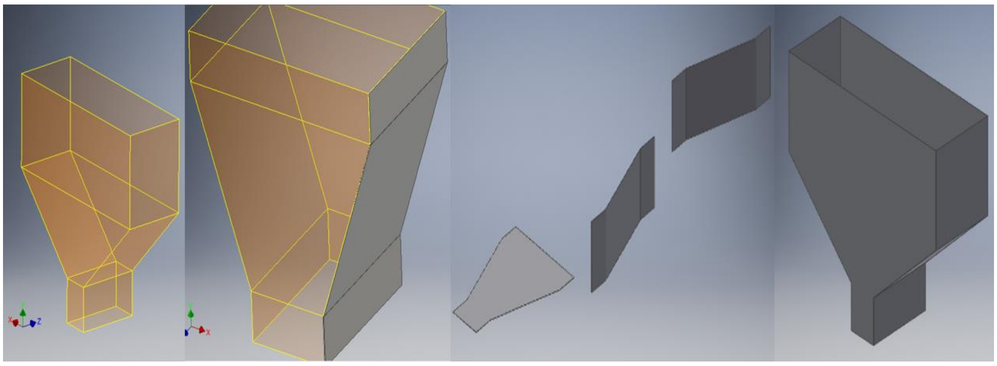

Figure 12. The grounding of the derived parts (MGFX, 2015)

4.6 **Multi-body part**

Multi-body modelling, as the name suggests, assumes the existence of several solid parts in one file which could be affected independently. Such conditions could be accomplished by designing features as different units what would create additional sub-folders in the software delimiting independent bodies. Patterns or mirrors could also be utilised to model solid bodies in the file. However, when utilising these features, the parts should be created in the form of bodies, not features or surfaces and should not be merged together what results in an opposite effect. If the master body that was used as an original geometry would be altered, changes will propagate across the subtracted bodies.

When parts exist together in the common file, they do not interact with each other meaning it is impossible to create any feature between them, for example, fillets. To do so, the bodies could be combined allowing the helpful technique to be used for the design. There are three functions that could be selected during the combination: common, add and subtract. (Aprigliano, 2013)

When two parts are modelled so that they have an intersecting area, a "common" function allows keeping the shared volume of two elements creating complex shapes what is represented in Figure 13.

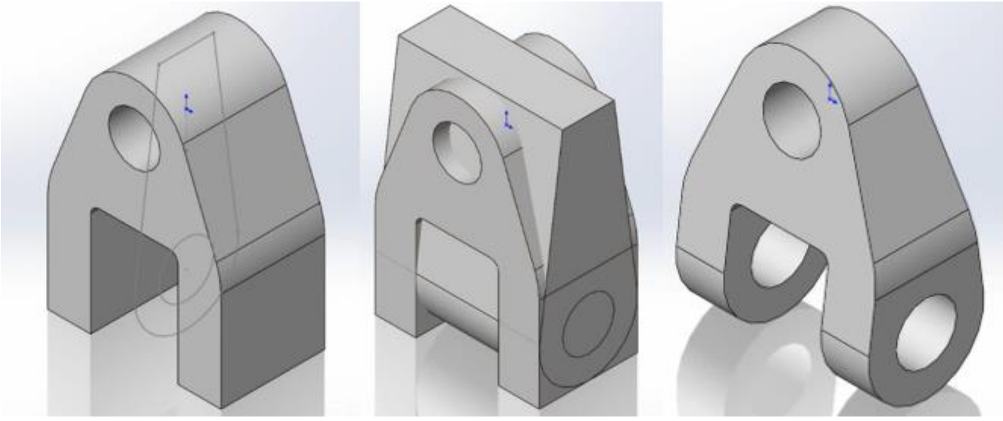

Figure 13. Combined bodies form a new part (Aprigliano, 2013)

“Add” function units all of the bodies into one making it a single part. All folders that contained the data about solids would be joint in one, negating the ability to work with each body individually. It is applied when there is no need to modify objects regarding the fact, they are detached solids.

In case of the need to remove one part from another leaving some geometry, “subtract” function could be used. Several elements are inserted in the working area, located so that they intersect in the correct place and then the subtracted element is selected and removed from the part leaving an indentation. An example is illustrated in Figure 14.

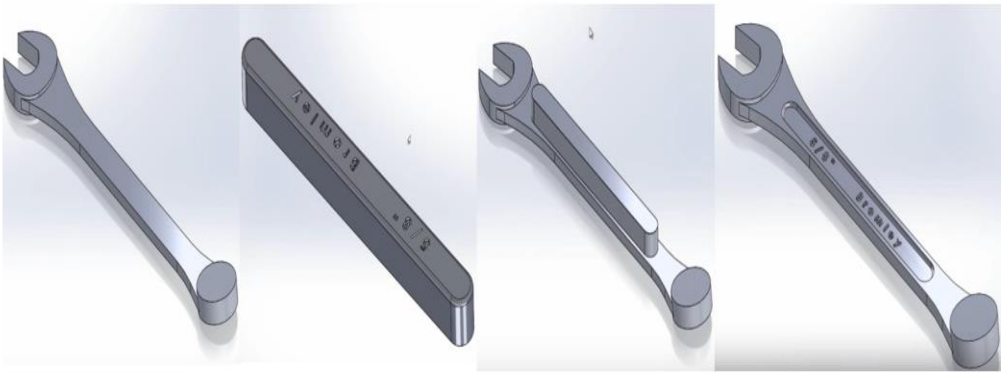

Figure 14. The subtraction between the combined bodies (Aprigliano, 2013)

When the multi-body is created in the file, a move-copy bodies function could be accessed. It enables designers to change the location of each solid element in the space similarly as it would work in the assembly relating them to other elements with various links. On the other hand, the kinematics of the parts is not functioning.

The multi-body could be successfully implemented in the project to substitute the assembly where its elements are not as crucial as the assembly itself or alleviating the file’s memory footprint. The technique is commonly applied when the modelling of the overall shape of the object comes first and then it is divided into several parts. An example of this could be a car-body modelling. (MLombard, 2013)

4.7 **Digital engineering design table**

Design table is a feature that combines the modelling process in the 3D software with a functionality of the Microsoft Excel. This tool allows applying mathematical formulas to calculate and manage parameters of the part which will automatically change the values to fit. The table itself is not an independent Excel file that can work autonomously but exists as a part of the model.

Digital engineering design table could be successfully implemented at different levels of modelling:

- Parameters
- Configuration properties
- Parts of the assembly
- Features of the assembly
- References

Before starting to add the information in the table, some actions should be made. Firstly, it is profitable to rename the dimensions in a way it could be easily understood by other people using common terms, such as width, height, diameter and others, to make an orientation among the parameters more convenient. Secondly, it is useful to show the names of the dimensions in the modelling environment by activating them from the annotation menu.

When working with a design table, the software opens an excel interface what is presented in Figure 15. All typical Excel’s operations are available in this mode. When the dimensions, names and their conforming values are selected from the part they are automatically transferred to the proper cells in the design table what minimise the amount of work an engineer should make. The dimensions that have been used for the initial part’s modelling, form a default setting. If they will be altered in either CAD software or Excel, the change will be automatically applied to another one correspondingly.

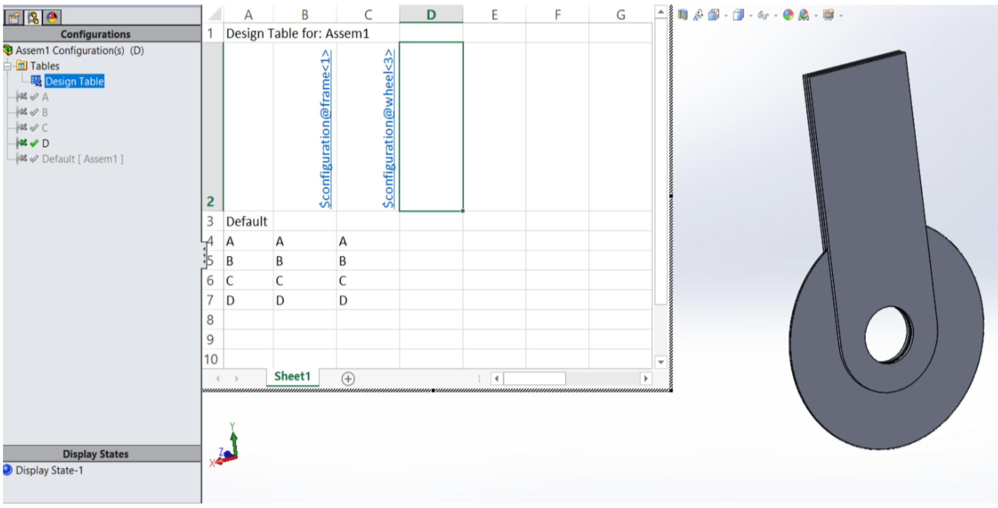

Figure 15. The design table interface (Ubaidullah, 2018)

The design table is mostly created in order to define various configurations of the part among which designer can easily switch without restating dimensions of features. They are made as additional arrangements of values in the table. The parameters of the features could be inserted not only with concrete values but also with the help of equations that build another relation between 3D modelling and mathematics. Additionally, a possibility to relate the configuration's values to other tables which might contain, for instance, standard parameters in order to exclude the necessity to insert them manually many times is a beneficial tool of the Excel.

When the design table is filled, the configurations automatically appears in the modelling software what empowers a convenient and quick shift between the characteristics to fit for different circumstances.

4.8 Horizontal modelling and the BORN method

The operations with overloaded assemblies could be complicated due to the big number of inserted objects that are loosely connected to each other and tend to disintegrate if some wrong actions are done. In order to have more reliable and stable models, horizontal modelling should be taken into consideration. Typically, most of the parts are modelled in the hierarchical manner where each next feature is referred to the previous one. Unlike it, the main concept of the horizontal modelling is to have all units of the system be referenced to the core. Two ways of 3D modelling differ significantly what could be observed from Figure 16. (Matt, 2010)

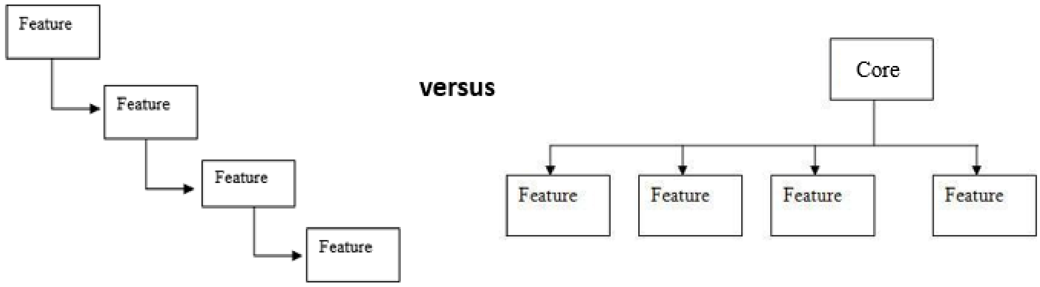

Figure 16. The comparison of the hierarchical and horizontal modelling

One of the common methods of the horizontal modelling is a BORN - Base Orphan Reference Node – that is founded on the idea to link objects to the origin reference XY, YZ, XZ planes, XYZ axes and a centre point of the origin which are illustrated in Figure 17. (Flayler M. A., 2008)

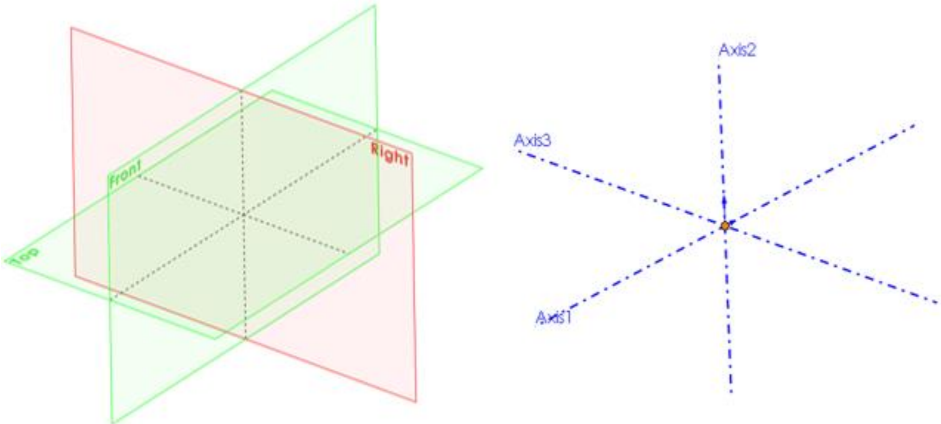

Figure 17. Primary planes, axes and an origin

These elements are the most reliable units of each model as they are rigidly fixed at their place defining the middle point and directions in the part file and could not be moved. Consequently, when utilising the BORN technique, it is essential to easily and accurately orientate in the modelling environment among different planes. A helpful colour-correlation prompt could be used in order to visually identify in which plane a part is created if they are not named; the colour of the origin's arrows corresponds to the colour model's name: XYZ = RGB. (Flayler M. A., 2008)

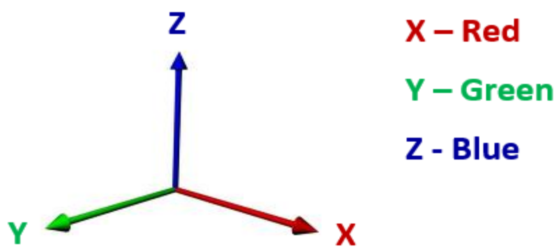

Figure 18. Correlation between the origin arrows and colour model's name

To control the model at a higher level, all the features that are designed on the primitive body should be linked to the origin or planes that have been referenced to it. The parameters should be assigned as projected or normal to geometry. Although the BORN method denies any external references which allow propagating changes across the parts automatically, a stadial approach in the mating of all elements to the core body results in a consistent and rigid structure that will not break in case of an accidental and significant shift.

5 STUDY CASE

To demonstrate capabilities of design approaches, the tutorial model - shipyard container - which could be observed from Figure 19 was offered by the third party. It was initially completed with the use of virtual parts which were fixed in place, but the attachment was done randomly in order to avoid any guideline for the following modelling. The part was created comprising various types of manufacturing and assembly methods and it represents the level of complexity of a standard medium-size project of the Company.

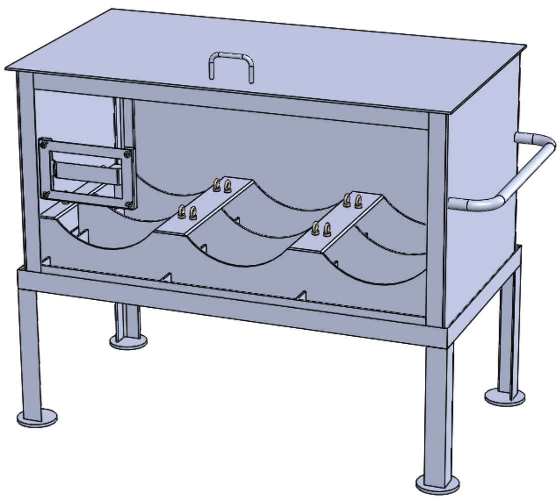

Figure 19. Original model with the front wall being hidden

The desired outcomes of the modelling are to build the part in the SolidWorks software with Bottom-Up and Top-Down approaches, get a similar product which acts differently due to the design intent and record what are the advantages and disadvantages of both methods. The focus of the research project should have been directed to the Top-Down as to the subject of study, however, the Bottom-Up should have been performed to ensure unbiased comparison from a different perspective.

5.1 Bottom-Up workflow

The Bottom-Up principles are going to be described shortly as they are widely known and traditionally used. Consequently, the only overview of basic issues is going to be presented.

Manufacturing of the product is a final goal for engineers. Therefore, despite of the selected design approach, a part should be modelled in accordance with the subsequent production requirements. The product should be divided into subassemblies from which drawings could be obtained. They are an essential part of the production as 2D technical drawing is the most common way of communication between designers and manufacturers. Therefore, manufacturing capabilities should be estimated and implemented into the design at the early stages.

The hierarchical order of parts comes from the sequence of the product’s assembly procedure. The offered model of shipyard box consists of three main parts: box, stand frame and inner holder. They could be seen from Figure 20. Those are the elements which are going to be assembled at the last stage to form the final product. Additionally, the box consists of several subassemblies which are going to be assembled at different stages.

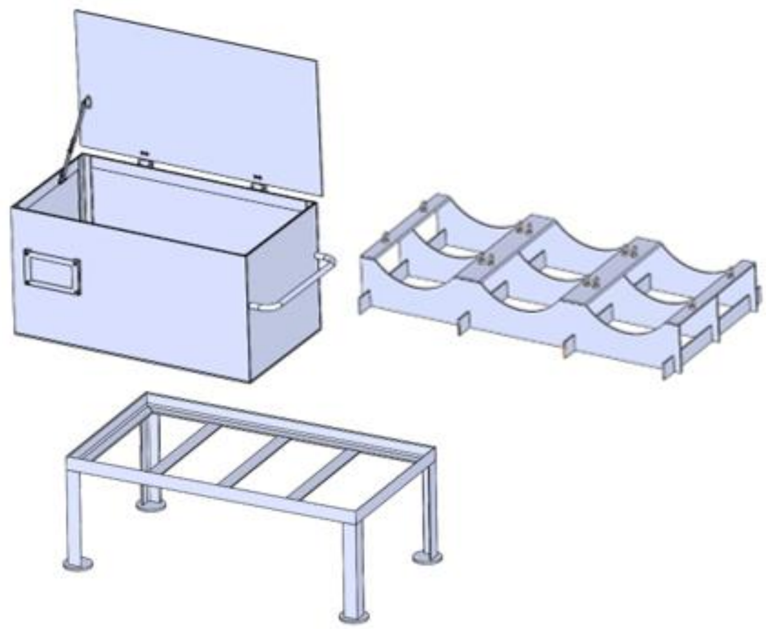

Figure 20. Components of the product

Technical drawings of products should include all necessary information presented in a clear and consistent way to ensure the existence of all required parameters for manufacturing. For this purpose, a part’s file should contain only the required elements in it as they are going to be visible in the drawing. According to the workflow procedure and production limitations, for example, several plates are going to be firstly welded together, then holes will be drilled, and fasteners will be attached. The order comes from the fact that welding causes many distortions. If holes are made in the beginning, the accuracy will decrease. The knowledge of manufacturing principles is important for engineers as it allows to design well-thought parts to which less alteration should be applied.

If to consider the overall Bottom-Up modelling procedure, firstly, each separate entity was modelled independently. Its dimensions are set with fixed values in the sketch which is used to transform it into 3D geometry. Secondly, parts are mated to each other at different hierarchy levels to create subassemblies with required technical drawings. The linkage is done basing on types of the relation between objects' surfaces what attach them to each other. Finally, the main assembly is created out of the subassemblies of the previous level with the insertion of parts which are added only on the final stage.

5.2 Top-Down workflow

The Top-Down design intent is a completely different approach which is based on accurate comprehension of the modelling process. The type of master part which will control the system should be selected, its efficiency should be ensured, necessary techniques be identified before any part is created and the type of mating of all objects be forethought. These steps are performed to minimise the number of alterations and, consequently, costs.

Even though design intents have differing characteristics, the division of the final product on subassemblies in Top-Down is done similarly to the Bottom-Up as the desired outcome is identical: correct technical drawings. However, the order of modelling varies dramatically. Engineers should define the shape and geometry of the product, possible relations between parts and functionality in advance. It is important because it will affect the modelling of the master element which will control the whole system. Moreover, it should be designed granted that essential features will easily be managed and that subassemblies might require specific features for their mating.

For the following assembly, several features were decided to have a priority: parameters of the shipyard box and relations between its elements. The reason for the selection of these criteria is a variety of container's sizes which might be required by the customer. The box is a standard product and it should suit different purposes and, consequently, its 3D model should be fully and easily altered with minimum efforts if such a need occurs. What is more, the mating of parts is a key factor in the modelling procedure. Since mates which are built directly between two elements, keep both parts at the right position, the elimination of one of them will delete the location foundation for another. To have better control over these parameters, the decision to use a skeleton which will be used as a basement controlling the whole assembly was made.

Overall, in the following modelling, next techniques have been tested to estimate capabilities of the Top-Down design intent in the SolidWorks environment: a skeleton, adaptive parts, layout sketches, the derive function and flexibility of parts.

The modelling started from the head sketch which is inserted into the main assembly. As Figure 21 illustrates, it is a rough 3D sketch which defines main dimensions and geometry which will be used as a reference for others. Only main parameters were captured to shorten the time spent on the element which would be probably changed dramatically before it would suit the purpose. The master sketch behaves as a driving element; therefore, it should include elements to which real parts will be linked and referenced to since they should behave as an adaptive geometry.

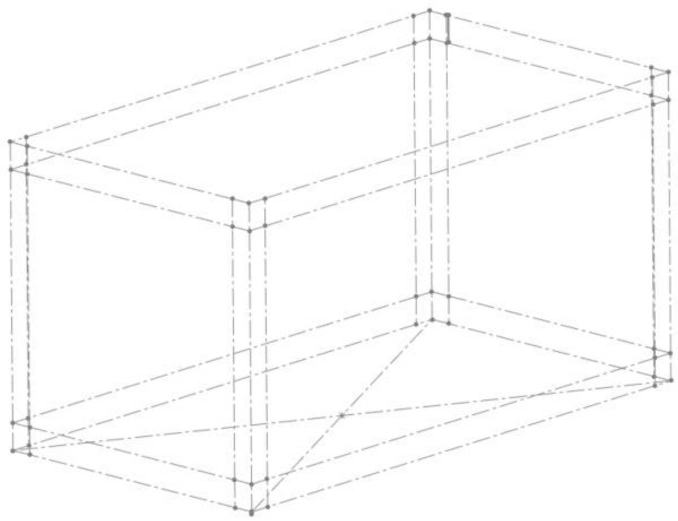

Figure 21. Rough master sketch

The master sketch should have all the necessary elements to be used as a reference for subassembly sketches. The whole sketch was firstly drawn with solid lines. When proceeding with building up a model, the sketch was modified several times by drawing additional geometry. For instance, offset rectangles which are shown in Figure 22 were added at the bottom to be used as location reference lines for the walls' attachment.

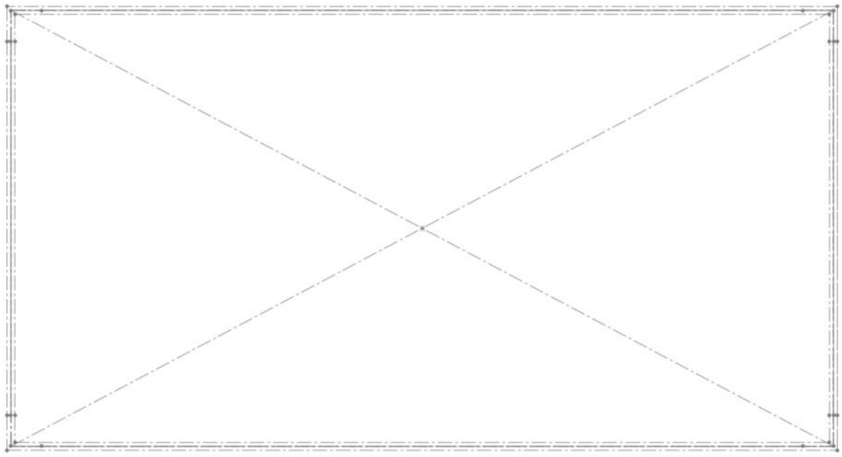

Figure 22. Offset rectangles of the master sketch

Afterwards, subassemblies' sketches were drawn. They accordingly should reflect the shape of the following subassembly. For instance, to manipulate the inner holder, its sketch was initially drawn inside of the master sketch which is shown in Figure 23 with solid lines. However, after the insertion of parts to the master assembly and attaching them by mating to the sketch, the problem has arisen: when trying to open parts as an independent subassembly, they were not fixed in space as mating was performed at a higher level. This information could be found from the design tree which is shown with (-) sign. The solution was not suitable as part should be locked in the space. Therefore, it was decided to add complementary stages in the process where each subassembly will have its independent sketch.

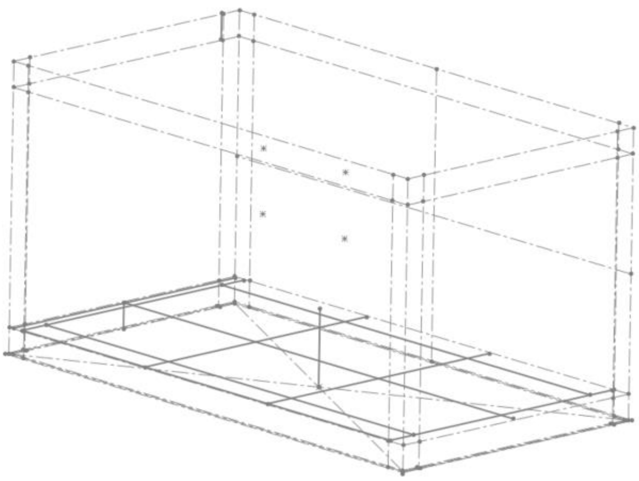

Figure 23. Subassembly's sketch drawn as a part of the master sketch

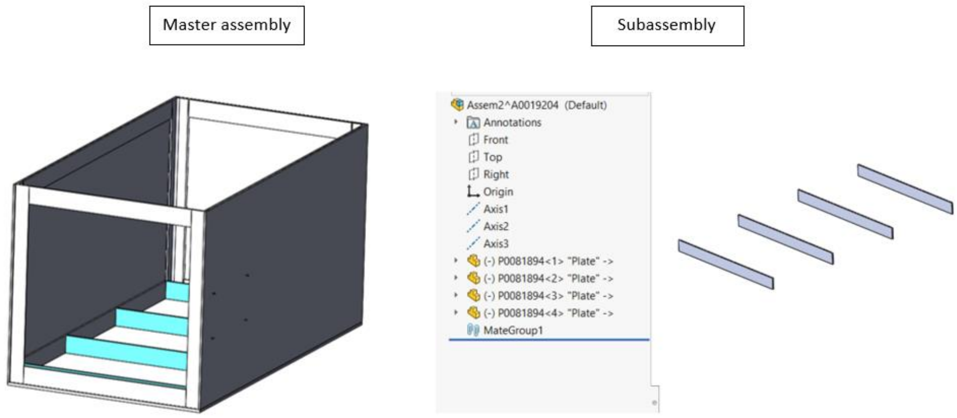

Figure 24. Floating elements

With the progress of the work, the complexity of sketches changed to satisfy requirements. Reducing the number of similar dimensions inside of one sketch is a beneficial practice as due to the correct relationship all similar elements will change accordingly when one is modified. What is more, to control the dimension of components, it was required to reference parts' sketch lines to be equal to the lines of a similar length in the master sketch. Under this circumstance, several longer lines were trimmed in certain places to build smaller sections what could be observed from Figure 26. The Figure presents the way how several elements of the sketch were related to each other to ensure correct propagation of modifications and trimmed to have a required length. Equal lines are shown with the same colour. However, if each line was dimensioned with separate value, then the Top-Down method would not work properly as it is supposed to. Additionally, the skeleton should contain geometry which will help to mate inserted components. For example, the inner holder subassembly parameters and location were controlled with the offset rectangle which was described above to make the sketch adaptive

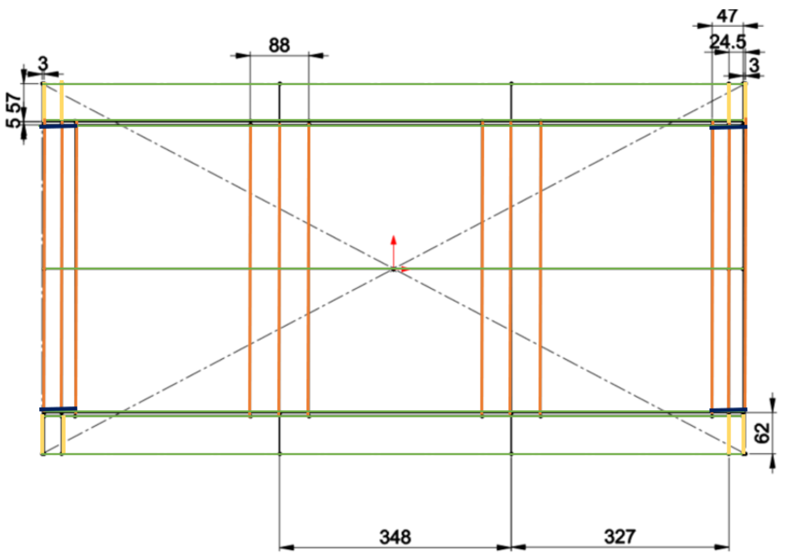

Figure 25. Reduced dimensioning in the sketch

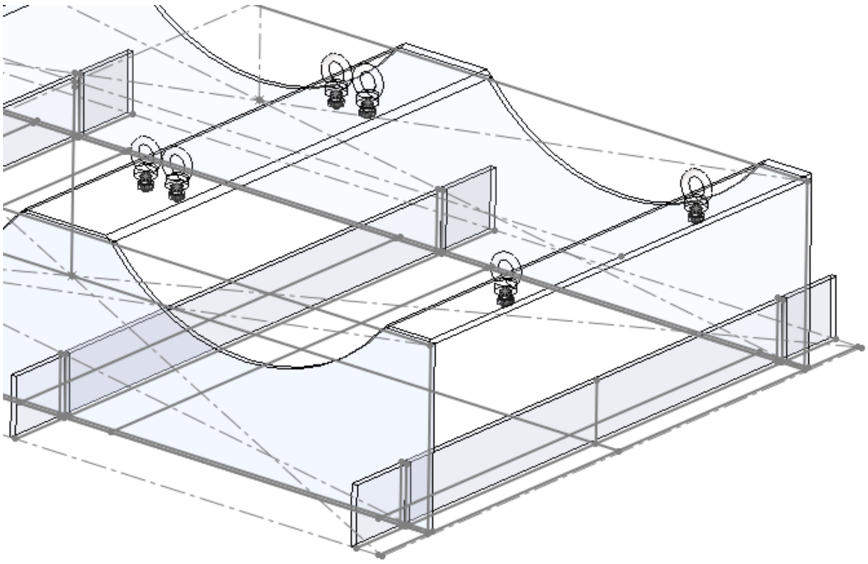

Figure 26. Sketch sectioning

The position of holes should be defined in the sketch to prevent shifts of linked components during the size's changes. Points were dimensioned being placed on the reference plane with fixed values relative to the outer frame geometry to be used as the centre point for holes and concentric mating for the information card holder as Figure 27 shows. The location of holes should not change in spite of the size of the box as card holder should stay at the same position.

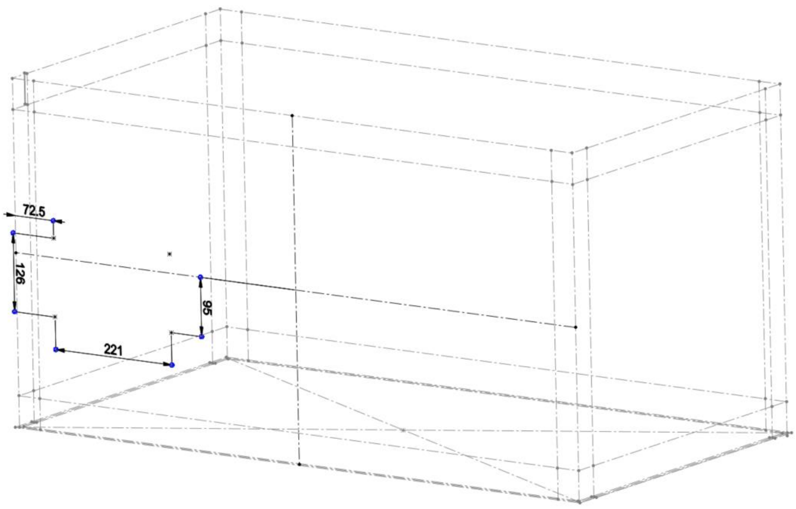

Figure 27. Fixed dimensioning of points

When subassembly sketches are inserted into the master sketch, their value dimensioning is deleted and external references with equality to the drive sketch are set. There are marked with the arrow "- >" in front of the part’s name. External references are typically avoided in bottom-up as they might accidentally build undesired geometry correlation but in top-down, this is one of the prime techniques. This procedure results in a strong connection between master sketch’s and subassemblies’ parameters meaning changes automatically propagate to all lower levels when the head sketch is modified. After the bonding actual parts were inserted from the initial model and fixed to the place in relation to skeletons. None of the parts could be connected to each other. Only part-skeleton mating is allowed. Moreover, it is essential to be aware of the type of relationships which are created. There is a chance to accidentally use in-place mating which alters the parameters of parts and save them being located in the exact place in space in reference to master sketch. However, as they exist in a certain location, it means it would be impossible to place a copy of the part correctly as it will break previous in-place mating.

Before assembling elements at the top level, each component was firstly tested for the ability to withstand modifications independently to reassure the proper behaviour. Figure 28 illustrates the example of changes applied to the outer frame’s sketch which have successfully propagated to the mated parts.

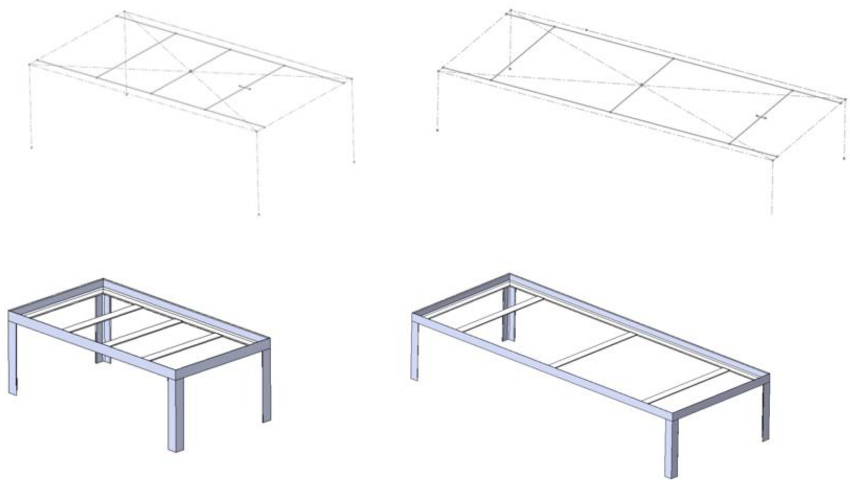

Figure 28. Comparison of the original and modified outer frame

After the mating of all components, the testing of the whole structure was conducted. The size of the assembly has affected the software performance which resulted in several crashes. To increase a work capacity the large assembly mode was activated. It hides all annotations, for instance, dimensions, planes, sketches and switches to the “view without edges” mode. It lightens the model and allows the software to concentrate the capacity on a calculation of modifications, not on the display. The first alteration was done in regards to sizing. The outer parameters of the master sketch were increased and decreased. Mostly all changes have propagated correctly. The problem concerning the inner holder arose. The reason for it was identified: while the size of the box changed, the number of radial trays did not change accordingly as they were not related. They were no more aligned with upper hooks plates when parameters altered. Several adjustments were applied to fix the issue; however, the solution was not found. The reason for it comes from the modelling of the plate from the initially offered container: radial shape tray was drawn at the sketch level but not done with a pattern. Therefore, it is not possible to control the number of trays with equations. This problem highlights the importance of accurate preparatory planning of the Top-Down modelling as certain issues might appear at the level when they could no longer be fixed.

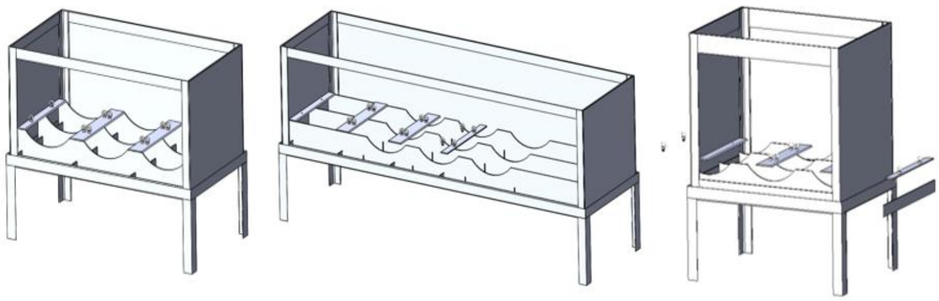

Figure 29. Test on the modification's ability

The shipyard box includes a moving lid which is attached to the box with hinges and a hatch. Such SolidWorks feature as a layout sketch was first selected as a possible solution to ensure the stable attachment of elements. Figure 30 shows both hinges and a hatch which were modelled by connecting blocks with the insertion point and transforming sketches into 3D geometry. The layout technique worked correctly independently, however, when being inserted into the assembly, the mechanics were blocked by the software. In order to solve the issue, the flexibility feature was tried to be applied which allows a part to move not just in their file but also when being copied to other locations. However, a layout sketch which prime feature is to perform steady kinematics was blocked at the assembly level. The part was forced to change to flexibility from the component property menu, however, it broke necessary relations and destroyed the layout assembly what is presented in Figure 31.

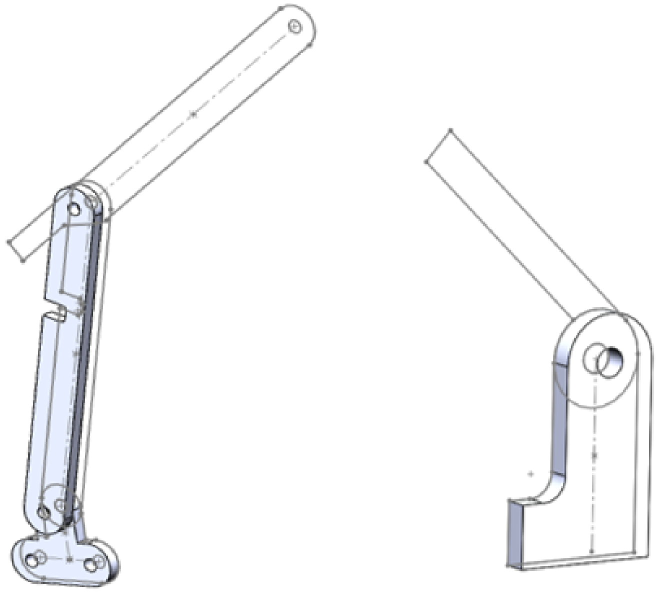

Figure 30. Hatch’s and hinge’s layout sketches

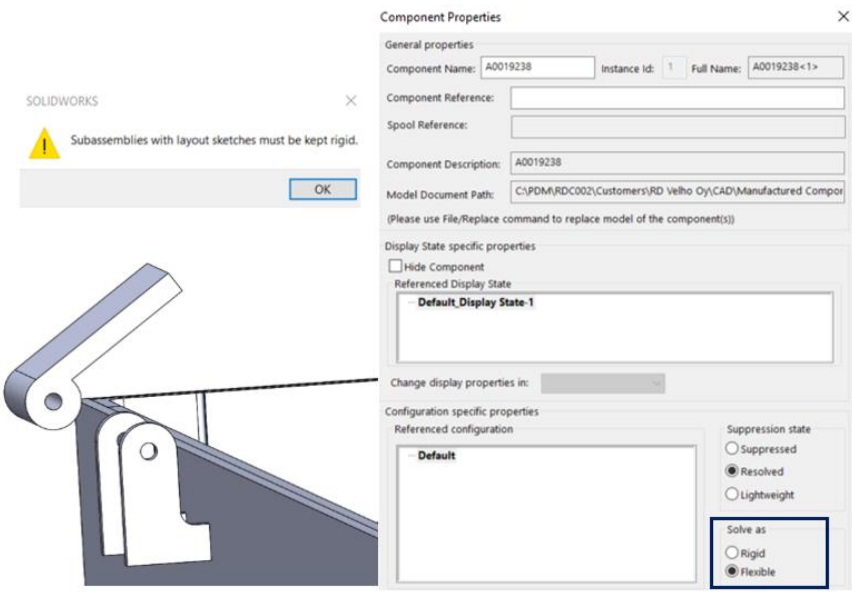

Figure 31. Flexibility feature breaks layout assembly

On this condition, the layout was substituted with the basic assembly. The location of hinges was defined with master sketch’s geometry. It has included a point which defined the centre of the rod and two elements of the hinge and two lines to control its location on the back wall. The sketch could be observed in Figure 32.

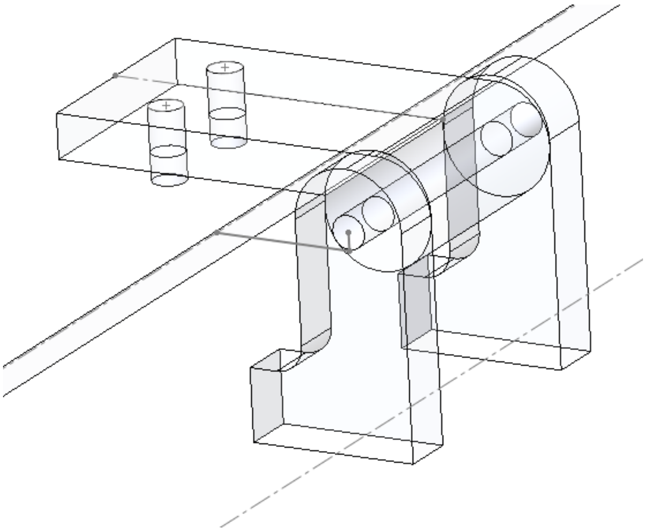

Figure 32. Alignment of the hinge with the master sketch

The lid was attached to the moving hinges and adjusted to be flexible what allowed free rotation at the master level stage. When the lid was mated to the box, an unknown problem arose. Even though the width of the lid was equal to the length of the drive sketch, the software was insistently changing dimensions with no obvious prerequisites. The issue is shown in Figure 33. Remodelling of the original lid saved from the offered model was conducted. Even though the initial sketch was modified, referenced to the skeleton to become adaptive and inserted into the master assembly, the issue kept arising with no logic sequence. The software did not notice the problem meaning no mates failed. Therefore, the solely functioning solution was to undo all of the steps done in the session to return to the moment where the size of the lid was correct.

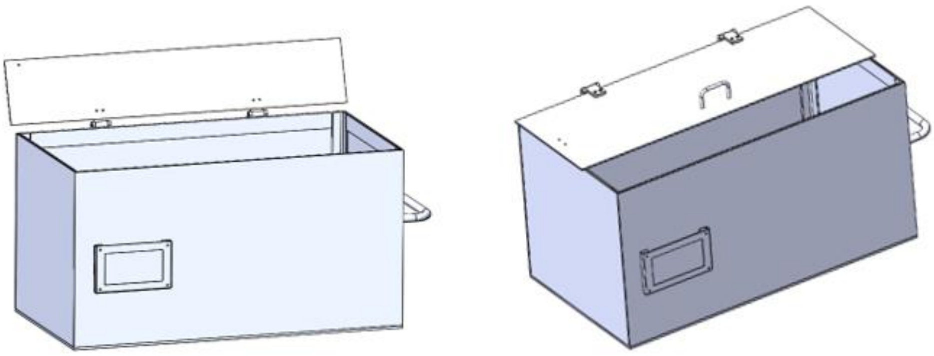

Figure 33. Broken mating between lid and master sketches

5.3 Final assembly

The final assemblies' results could be observed in Figure 34. As it could be noticed, both models look similar as a design intent does not affect the final product appearance. Moreover, parts are divided into similar subassemblies due to the fact that it is going to be manufactured with the same sequence. The design approach has an effect only on the modelling workflow. Therefore, the modelling tree of the Top-Down part includes additional elements – skeleton sketches – for each subassembly. The figure illustrates the simplified comparison diagram of Bottom-Up and Top-Down model trees.

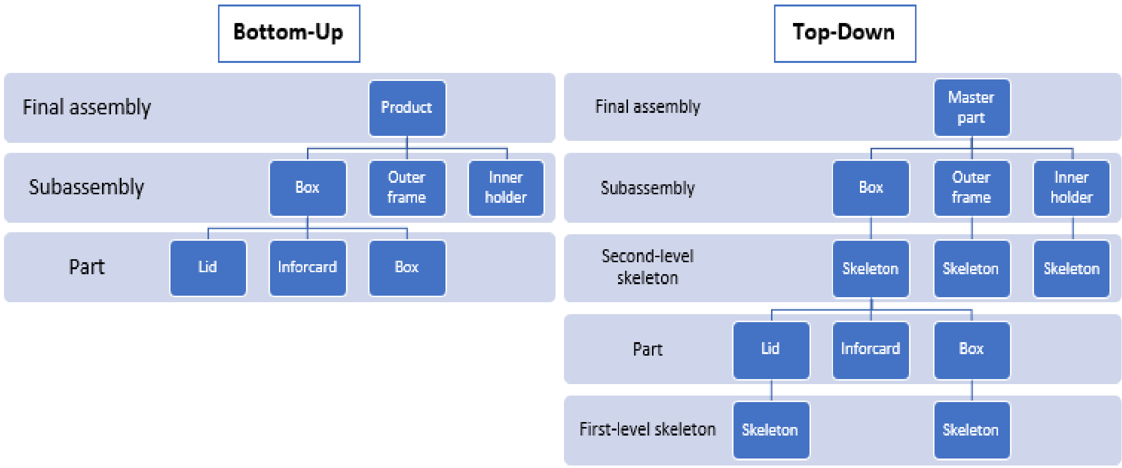

Figure 34. Simplified comparison of the Bottom-Up and Top-Down hierarchy trees

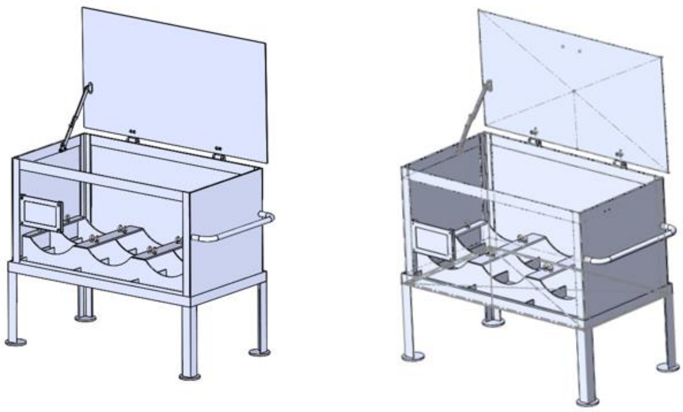

Figure 35. Bottom-Up and Top-Down models

The additional difference could be observed on the mating level which approaches control. Bottom-Up components of the part are positioned with a reference to other elements around it. Each part has several mates to close elements what blocks its movements. However, assembly, while being consistent when all components are kept untouched, can be destroyed easily as soon as some elements are changed or deleted. On the contrary, all parts of the Top-Down assembly should be mated solely to the skeleton. It ensures that no breakdown occurs when several parts are removed or modified.

6 CAPACITY OF THE TOP-DOWN MODELLING IN STANDARD PROJECTS

6.1 Special characteristics of standard projects

In order to evaluate whether the Top-Down approach could be beneficially utilised on an ongoing basis in the Company's environment, it is necessary to estimate what are the special characteristics of the organisation's standard projects.

The Company is specialising in the sphere of service where a lot of effort is contributed to the development of customers' product. The work comes from the clients' orders who are aiming to create new products to fit their needs or to upgrade the existence items. Therefore, the Company's routine is mostly focused on the conceptualising, modelling, prototyping and preparing technical documentation of not standardised objects. All the designs are made in spite of the customer's challenges in order to solve their problems.

The level of product complexity varies dramatically from small measuring devices to enormous agricultural machinery meaning the Company works with all-sized projects. The number of product's component differed from 40 to 1050 parts in the assembly. Most of the products have complex compound shape geometry which is done with the surface forming. Also, there are no restrictions concerning the following production methods of the parts. Projects typically concern surface modelling, sheet metal, welding, piping, electronics, heating ventilating and air conditioning, specific electronic devices. However, the Company is not specialising in large scale transport, for instance, ships, planes, trains, construction engineering, gadgets such as TV's, phones, computers.

The Company's service payment is built upon the number of hours which is spent on the project. Typically, it is not restricted to the fixed value but is constantly negotiated with the customer who should always confirm the beginning of each new stage of the work. Time variation fluctuates strikingly: from 25 to 8000 hours were set for the work regarding the difficulty of the task.

When the observed data was summarised, the next similar characteristics of the standard project were detected. The most noticeable common denominator of the projects is diversity. All projects have different duration, a number of part and manufacturing methods. Additionally, all of the products are customer-specific.

6.2 Advantages and disadvantages of Bottom-Up and Top-Down approaches

Bottom-Up and Top-Down paradigms are commonly applied by engineers while modelling process. As it was proved in the study case, similar results could be achieved when utilising any of them. However, the practical application of the modelling approaches in SolidWorks software allowed to distinguish the strengths and weaknesses of methods.

6.2.1 Bottom-Up approach

Advantages:

- Stable performance

  When the 3D software has fewer calculations to perform, it provides users with more consistent performance. It is an important factor while working with heavy documents, as unexpected crashes of the system without the ability to restore the progress might affect the work process dramatically.

- Fast accomplishment rate

  The Bottom-Up approach does not assume the necessity to create any additional geometry for part's placing and mating. Consequently, the modelling process is straightforward and close to real-life manufacturing.

- Short training

  Typically, when a person starts working with 3D modelling, the Bottom-Up technique is taught firstly as assumed to be easier for perception and comprehension. Hence, it will not be required to spend time on employees' teaching before the start of the work.

Disadvantages:

- Hard to modify

  Each of the assembly's parts is modelled separately when utilising Bottom-Up approach. Most of the elements in the system do not have any relations in-between each other apart from the mates. Therefore, if modifications should be applied to the part, it would require updating the parameters of each element independently what is time-consuming.

6.2.2 Top-Down approach

Advantages:

- Capability to withstand modifications

All parts, which might be modified in the model, are mated to skeletons to avoid any referencing to other inserted components. Therefore, no mating mistake occurs if the skeleton sketch is initially was drawn correctly and will not be subjected to alterations. Since it is kept untouched or the modification will not affect crucial geometry, the overall assembly will not break down due to the removal of several parts or change of their dimensioning or even shape.

- Greater control over the complex assembly

  The number of inserted components might exceed thousands in certain cases. Thus, more elements assembly has, easier it brakes because of a big number of various parts' bonds which are hard to control. However, the existence of a master part which drives the whole structure helps to manipulate the assembly and its elements in a more reliable and robust way.

- Multi-users

  Due to the fact that the Top-Down modelling approach assumes the utilisation of the managing part which is modelled firstly and then divided into smaller subassemblies, it is possible for several people to conveniently work on the subassemblies. When their skeletons are inserted into the shared master part and positioned, they reserve the space which could not be occupied by other people. Moreover, as they update their subassemblies, changes automatically spread to other levels allowing the simultaneous work.

Disadvantages:

- Large amount of preliminary work

  The Top-Down modelling is impossible without accurate planning of the work before the start of the project. Many aspects should be taken under considerations to ensure proper functions' execution. The SolidWorks does not provide users with auxiliary features to facilitate the working path. Therefore, each step should be forethought to fit the purpose of the work to eliminate the chance of a complete remake.

- Compulsory implementation from the beginning

  The decision about the type of the design approach which is going to be used should be made beforehand as a switch from one type to another might be problematic. Each method requires a specific way of components' handling and parts' maintaining, therefore, they should be treated correctly from the beginning to result in the required outcome.

- Time-consuming

  Firstly, preparatory work requires a certain number of hours to be spent. Secondly, such additional elements as, for example, skeletons or design tables, are completed within working hours. Thirdly, it might be challenging to develop the master part correctly right out of hands. It will be subjected to many modifications during the work progress. Hence, frequent return to the modelled part to make adjustments results in the increased time spent on the product.

- Unclear handling

  The Top-Down sketches are completed in a manner where similar dimensions and geometries are transformed to be equal or correlate with each other. This is done to ensure phased propagations of changes which affect all elements simultaneously. However, this type of referencing hides many dimensions what makes sketch harder for understanding. Consequently, un unclear visualisation of the parameters might cause issues with understanding when new members join the project in the middle of the process.

6.3 Applicability of the Top-Down approach in the standard project

During the progress of the study, two types of design intent were tested for the purpose of assessing the level of Top-Down approach's fitness to the aim of the Company: efficient modelling of products with minimum risks and costs, maximum output and quality. To summarise data collected from the theoretical study and practical application of knowledge, the following evaluation of the method's capacity and applicability was conducted.

As it was previously elaborated, the main characteristic feature of the common task in the studied Company is a versatility which assumes regular and cardinal shifts between types of projects. Hence, it results in the impossibility of the creation of concise guidelines which should be strictly followed by employees during the workflow. Each of the commissions should be handled in a unique way to ensure efficient utilisation of the resources, achieve the highest end result and meet client's expectations. Under these circumstances, the implementation of the Top-Down method in SolidWorks projects could be challenging as it requires a high level of the mindfulness during the modelling process which could be hard to achieve without any experience in the employment of Top-Down's specific techniques. Thus, it will result in an increased number of non-profitable hours for the company spent on training.

The concept stage in the process of product development includes next requirements: satisfy the terms of the commitment, deliver the high-quality product and technical documentation and fit in the stipulated time limit. Due to the large workload and a big number of simultaneously on-going projects, the work should be performed quickly with a minimum number of errors and delays. In the case of the particular company, first outcomes from conceptualising and prototyping stages should not be too detailed as the product might withstand numerous alterations before being finalised. The Top-Down approach, however, might lead to the additional preparatory work, for instance, building skeletons, which should be completed beforehand to ensure the correct functioning of the structure. Top-Down's assisting techniques will inhibit the work rate.

Although the preliminary preparations for the work expect all the steps to be processed ahead, in reality, many shortcomings might be committed. Additional time will be spent on the testing of the stability and reliability of master elements and selection of correct tools. For example, the skeleton requires to be continuously updated to satisfy its purpose. Furthermore, the pre-made part might be subjected to complete remodelling or even deleted on the conditions of dramatical changes in the prototype. In spite of the fact that the income of the reviewed Company comes from the completed project, or, specifically, the number of allocated hours, extra work is unwanted.

Many of projects concern the invention of narrowly aimed products which will exist in the single configuration which will not be resized or changed. The components of the assembly are located and mated once. Therefore, Top-Down methods which aim to increase the level of adaptability to modifications will superfluously increase the complexity of models.

Even though the Top-Down could not be fully and strictly implemented in the standard workflow of the Company in spite of the described conditions, it is important to realise that separately its methods could become an applicable part of the modelling procedure. Each of the technique might be efficiently utilised in specific cases where certain capabilities of the Top-Down approach are required. For instance, the skeleton referencing might be substituted with planes, axes and centre point. As it was previously discussed in the chapter concerning the horizontal modelling, these elements of the part are evaluated to be more stable than skeleton as they cannot be modified. Therefore, components of the assembly might be linked to them instead of the sketches. Although the dimension and geometry management will be lost, a fixed positioning of the parts could be assured.

Additionally, the impact of the modelling environment in which the method was tested should not be omitted. The capacity of the software limits the performance of the used design approach. The potential of the approach could not be entirely achieved when the programmes are not capable to support the level of complexity which is created in the model. Therefore, it should be stated that with the further development of 3D CAD software the efficiency of the Top-Down method's implementation could be increased.

Lastly, the opportunities for further research in the field of Top-Down modelling should be mentioned. Due to the restricted duration of the project, the area of the study was narrowed down to reach solid outcomes. In spite of this fact, only certain Top-Down methods were tested limiting the scope of the existing resources. Therefore, the subsequent studies might discover more suitable techniques which could be successfully implemented into the standard procedure of the Company.

7 CONCLUSION

Nowadays three-dimensional computer-aided design software is the main tool of designers and engineers which is used for the creation of the volumetric models in the virtual reality. Due to the complicated mathematical and geometrical calculations which are performed on the background of the modelling process, people have been enabled to create realistic objects which can identically represent the parameters of the real physical item. In order to model complex assemblies, all elements of the final product should be positioned in the space in relation to each other or origin reference elements. To guarantee the best performance of the model, it should be made in a coherent manner with an accurate understanding of the theory behind the procedure and structured plan of the development stages.

The selection of design intent plays a significant role in the determination of project success. Different types of modelling approaches focus on the mating of the structural components of the assembly and define the manner in which modification's affect the model. While being developed due to the diverse attitudes to the task managing and supplying users with unlike techniques for the implementation of the philosophies to ensure the consistency of the workflow, outwardly similar product could be achieved with the utilisation of each of them. Instead, the inner structure of the modelling is a varying parameter which is driven by the design intent.

Overall, such characteristics of the project as its duration and scale have a critical impact on the selection of the design intent. Based on the research case which was conducted to test the methodologies of design intents in close to realistic conditions and designate the capacity of the Top-Down approach in case of the Company's standard project.

All things considered, the capacity of the Top-Down approach is evaluated to be limited in the circumstances of the studied Company in comparison with the Bottom-Up. Its potential could not be entirely unleashed provided that the best condition for the method's application is a standard part's project which consists of a big number of fragments which would require to be altered often. The diversity being a common denominator for the standard Company's projects do not allow to use the approach properly. Top-Down philosophy brings undesired complexity to the model and requires a lot of time to be allocated for the extra preparatory work which might not bring any value afterwards. However, Top-Down could become a helpful tool for the large assemblies and its single technics be applied on a regular basis as auxiliary tools.

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